Faculty of Science and Technology

MASTER'S THESIS

Study program/ Specialization: Petroleum technology/Production	Spring semester, 2011 Open
Writer: Espen Myklatun Tveit	………………………………………… (Espen Tveit)
Faculty supervisor: Rune Time and Steinar Evje External supervisor(s):	
Titel of thesis: Simulation of a gas kick	
Credits (ECTS): 30	
Key words: Gas kick Two-fluid model Drift-flux model AUSM	Pages: 61 + enclosure: Program on CD Stavanger, 13.06.2011

Frontpage for master thesis
Faculty of Science and Technology
Decision made by the Dean October 30th 2009

Abstract

In this paper a gas kick is simulated using a MatLab program for the purpose of showing several important phenomenons that occur during a gas kick. Pressure, flow rate in and out, fluid velocities and fluid fractions is studied to shed light on the fluid mechanics behind these phenomenons.

The base theory of the drift-flux model is explained based on the book by R. Time[1], for the purpose of demonstrating how a drift-flux model work before it is implemented in to the full MatLab scheme.

Based on the work done by Evje and Fjelde[2] the two-fluid model together with the assumptions made in the model is explained. A hybrid of the flux-difference splitting scheme and the flux-vector splitting scheme is used in the model, based on AUSM and Van-Leer respectively. The reason of selecting this hybrid scheme is to be able to capture the fast propagating acoustic waves from the FVS scheme while avoid numerical dissipation by using the AUSM scheme.

MatLab setup for the model is explained in detail and who the different routines work together to make the model work. The main routine together with the sol_stian routine is given the most attention since these two routines are the biggest and most important in the model. After it has been established the function of every routine and how they work together in the model, it is explain where data must be changed in order to get the desired results from the model. A basic case is run to display how model function and what results the model produces.

The simulation is motivated by the paper written by Avelar et al[3] where the pressures in the well and fluid flow rates are simulated. This case will be referred to as the reference case in this paper.

In the results from the simulation it is seen that model produces the phenomenon one would expect from a gas kick in a well. When correlating the results with the reference case it is seen that graphs the simulation yields can be assumed to be accurate, since they have about the same shape and characteristics. Hence the model and assumptions made to implement the case in the model is accurate.

Acknowledgements

I would like to thanks Rune Time and Steinar Evje for spending their time to guide and help me trough this project. This thesis would not have been possible without their insight on the theoretical foundations and workings of the model. I would also like to thank my father Kjell Ove Tveit for spending the time to proof read this project.

Table of content

Introduction

When a gas kick occur during drilling it can be one of the most dangerous challenges that the drilling crew is facing. The potential damage to equipment, environment and the personnel is huge. To be able to prevent this from occurring and knowing how to react when it occurs is vitally important when drilling, especially offshore where leakage of oil into the sea can cause environmental disasters.

The main purpose of this paper is to simulate a gas kick in a well in order to get acquainted with the phenomenons that are displayed. Mainly the pressure changes in bottom and top of the well, inflow and outflow of liquids in the well and velocity of the fluids in the well will be studied. The simulations done by Avelar et al[3] will be used as a reference case to determine is the simulations done are sound, and the theory written by Time[1] and Evje and Fjelde[2] will form the basis of the model.

Modeled in MatLab will be a two-fluid hybrid model between a flux-vector splitting scheme and a flux-difference splitting scheme. The selection of a hybrid between the FVS and FDS schemes are motivated by the desire to create a model that give correct values for the propagation of acoustic wave while still not suffering from excessive numerical dissipation. Due to the complexity of the two-fluid model it will be simplified as a drift-flux model, selecting the drift-flux model is prudent in this type of two-fluid calculations since it is able to relate the velocity of the gas to the velocity of the liquid.

After constructing a model that is able to simulate the desired phenomenon in the well, all of these phenomenons will be explained based on multiphase flow theory, in order to shed light on why these phenomenons occur, how they occur and how they can be negated to avoid accidents while drilling.

Basic multiphase theory

Basic definitions

In order to do the complex calculations to predict the two-phase behavior, some basic definitions are required. These definitions will serve to highlight the fundament on which the complex model is based upon.

Consider a pipe with the area equal to A. In this pipe both liquid and gas are flowing, and the volumetric flow rates of the gas and the liquid are denoted q_G and q_L respectively. From these volumetric flow rates q_G, q_L and the area A of the pipe, the superficial velocities U_{GS} and U_{LS} can be defined:

$$U_{GS} = \frac{q_G}{A} \quad (1)$$

$$U_{LS} = \frac{q_L}{A} \quad (2)$$

The superficial velocity is in other words the volumetric flow rate of the phase divided by the total area of the pipe. This is not to be confused with the actual phase velocity u_G and u_L which is the volumetric flow rate divided by the area that this phase occupies in the pipe, A_G and A_L.

$$u_g = \frac{q_G}{A_G} \quad (3)$$

$$u_g = \frac{q_G}{A_G} \quad (4)$$

The area that one phase occupies in the pipe at any given time, can be written as the total area of the pipe multiplied with the fraction of that phase:

$$A_G = A \cdot \varepsilon_G \quad (5)$$

$$A_L = A \cdot \varepsilon_L \quad (6)$$

Since there are only liquid and gas in the pipe, these two areas will be the sum of the total area of the pipe. From these formulas it follows that phase velocity is equal to the superficial velocity divided by the fraction of the phase:

$$u_G = \frac{U_{GS}}{\varepsilon_G} \quad (7)$$

$$u_L = \frac{U_{LS}}{\varepsilon_L} \quad (8)$$

In order to link these two phase velocities together a new parameter called slip is introduced. This parameter is a dimensionless parameter and is defined as follows:

$$S = \frac{u_G}{u_L} \quad (9)$$

Fraction of the phase can then be written:

$$\varepsilon_G = \frac{A_G}{A} = \frac{A_G}{A_G + A_L} = \frac{\frac{q_G}{u_G}}{\frac{q_G}{u_G} + \frac{q_L}{u_L}} = \frac{q_G}{q_G + S \cdot q_L} \quad (10)$$

$$\varepsilon_L = \frac{A_L}{A} = \frac{A_L}{A_G + A_L} = \frac{\frac{q_L}{u_L}}{\frac{q_G}{u_G} + \frac{q_L}{u_L}} = \frac{q_L}{\frac{1}{S} \cdot q_G + q_L} \quad (11)$$

Further divide with respect to A in the nominator and denominator in formula (10) and (11), and from the relations in formulas (1) and (2), the fraction of the phase can be written as:

$$\varepsilon_g = \frac{U_{GS}}{U_{GS} + U_{LS} \cdot S} \quad (12)$$

$$\varepsilon_L = \frac{U_{GL}}{\frac{1}{S} U_{GS} + U_{LS}} \quad (13)$$

Thus the fraction of a phase can be determined by only knowing the volumetric flow rates and the slip between the phases. The slip ratio is connected to the flow regime in the pipe and will change based on the regime. Typically the slip will be approximately one for dispersed bubble flow and slightly over one for slug. However it may vary greatly for the stratified flow regime.

Drift flux model

In the drift flux model the reference frame is moving along with the fluid. The speed of the moving reference frame is the mixture velocity, which is the sum of the superficial velocities:

$$U_{Mix} = U_{GS} + U_{LS} \quad (14)$$

The two fluids in the pipe will then move forward or backwards relatively to this reference frame, or in the case of no slip move with the same velocity as the reference frame.

For the drift flux model in one dimension, it is assumed that the distributions of the fluids are equal over the cross-section of the pipe and that velocities of the fluids are equal over the cross-section. This simplification means that it is possible to set up two sets of equations which describe the velocity and fraction of the fluids throughout the pipe. These two equations are called the kinematic equations, and relates to the velocity, and the dynamic equation which relates to the properties of the fluids and their interaction.

First the kinematic equations for the liquid and the gas are defined as follows:

$$u_{GM} = u_G - U_{Mix} \quad (15)$$

$$u_{LM} = u_L - U_{Mix} \quad (16)$$

This is the actual speed of the fluid relative to the mixture velocity, however since the actual velocity depends on the fraction, it is more prudent to define a superficial parameter for the relative speeds. These superficial relative speeds are therefore defined as:

$$j_{GM} = \varepsilon_G \cdot u_{GM} = \varepsilon_G(u_G - U_{Mix}) = U_{GS} - \varepsilon_G \cdot U_{Mix} \quad (17)$$

$$j_{LM} = \varepsilon_L \cdot u_{LM} = \varepsilon_L(u_L - U_{Mix}) = U_{LS} - \varepsilon_L \cdot U_{Mix} \quad (18)$$

Since we know the volumetric flow rates of the two fluids, the only unknowns are the fraction and superficial relative speed. These two equations will, since the reference frame is defined based on the superficial velocities, sum to one. The fractions will also sum together to one, which means that only element to be calculated is the gas fraction.

Since there are two unknowns, two equations are needed in order to solve it. The second is the dynamic equation, which is an equation that relates the j_{GM} to the fluid properties, the fluid parameters and the flow regime in the pipe. The borders of this equation is when ε_G is close to

zero and when ε_G is close to one. When it is close to zero the mixture velocity approaches the liquid, and the gas velocity will approach the rise velocity of the gas. The rise velocity is the velocity the gas will have if the liquid is still and only buoyancy moves the gas. Based on these border conditions it is proposed that the following equation will give a good estimation:

$$j_{GM} = \varepsilon_G(1 - \varepsilon_G)^n \cdot U_T \quad (19)$$

Where U_T is the terminal rise velocity and n is a parameter which is dependent on the flow regime, but typically is around one half up to two.

These formulas can be written in to Matlab to create a program which simulates the fractions of the phases. The program that has been created can be seen in figure 1below.

```matlab
1 -   clc
2 -   clear all
3     %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
4     %%%%%%%%                 Input Data                 %%%%%%%%
5     %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
6 -   up      = 1;
7 -   down    = -1;
8 -   Gas     = up;
9 -   Liquid = down;
10 -  Uls     = 0.1;
11 -  Ugs     = 0.5;
12 -  Ut      = 5;
13 -  n       = 2;
14    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
15    %%%%%%%%                Calculations                %%%%%%%%
16    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
17 -  Eg      = 0:0.0001:1;
18 -  Ulsr    = Uls*Liquid;
19 -  Ugsr    = Ugs*Gas;
20 -  Umix    = Ulsr+Ugsr;
21 -  jgkin   = Ugsr-Eg.*Umix;
22 -  jgdyn   = Ut.*Eg.*(1-Eg).^n;
23 -  plot(Eg,jgkin,'b')
24 -  hold on
25 -  plot(Eg,jgdyn,'r')
26 -  hold on
27 -  xlabel('Gas Fraction')
28 -  ylabel('Jgm')
```

Figure 1

In this program the direction of the liquid and the gas is defined as either up or down in the input section. The superficial velocities are also defined, the terminal velocity of the gas and the constant n. Keep in mind that values for the terminal velocity and the constant n will vary along the pipe, due to the fact that the pressure will vary and in turn change the flow regime and the flow parameters.

9

For this simulator there are four different cases that can be run for the direction of the gas and liquid, however only three of the cases will yield an actual result due to buoyancy. The gas is unable to flow down if the liquid is flowing upwards. The three remaining cases will be simulated with $U_{LS}=0,1$, $U_{GS}=0,5$, $U_T=5$ and $n=2$ to demonstrate the model.

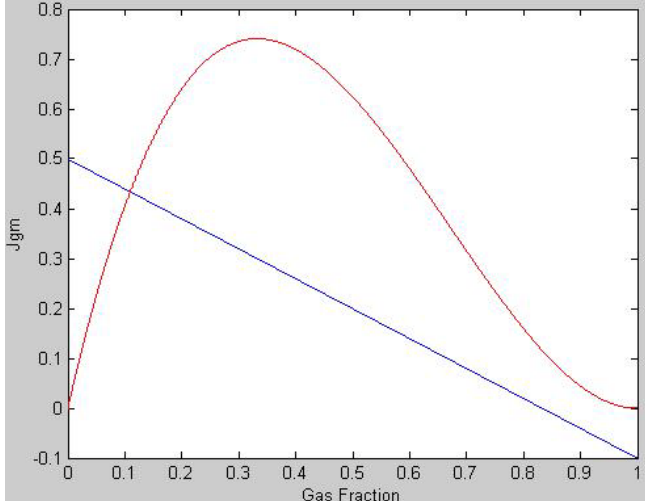Liquid up and gas up case results in the following graph:

Figure 2

There is only one solution for this case with a relatively low value for the gas fraction, as to be expected since the gas in this case will "gain velocity" from both the fast that it is co-current with the fluid and from the buoyancy. With the increased velocity and a constant volumetric rated being pumped in to the pipe it follows that the area the gas occupies will be small relative to the liquid.

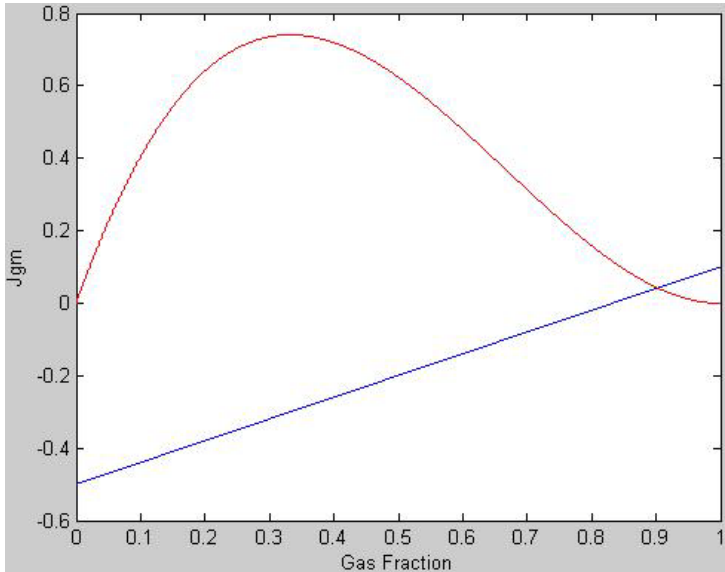The case of liquid flowing downwards and the gas flowing downwards yield this graph:

Figure 3

10

The gas fraction here is dramatically increased compared to the upwards co-current, almost 90% of the pipe contain gas. The reason for this is that the buoyancy is slowing the gas down rather than speeding it up like in the previous case.

The last case is liquid flowing downwards and gas flowing upwards:
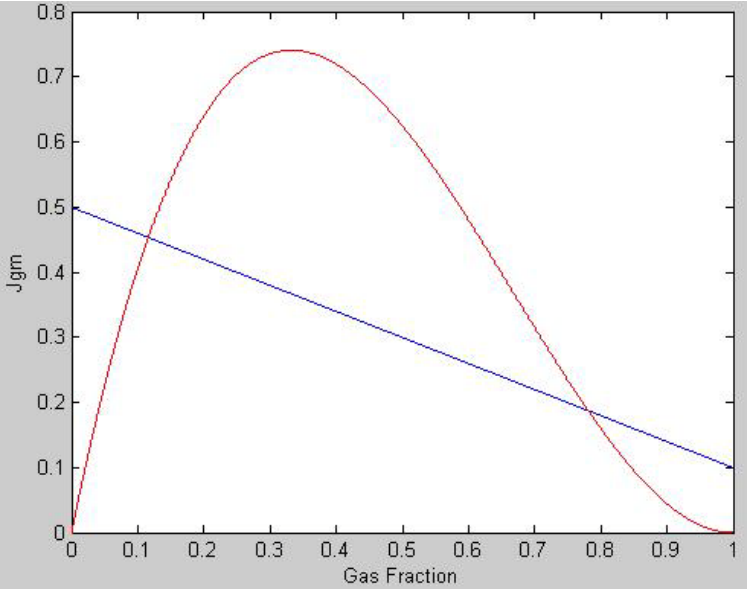
Figure 4

In the simulation there are two solutions present, and the correct one is depending on the size of the bubbles flowing in the pipe. However with a variation of the input data one could get one result or even none. If none results are present this means that flooding limit has been reached and the fluids are not flowing like the input suggest it is. One cannot have gas flowing upwards with liquid flowing downwards at very high speeds.

Theoretical basis of the simulation program

The program is based on a two-phase model for the fluids, where the changes in mass are calculated for the liquid and gas and the total momentum is calculated. Rather than using a model that simplifies this by the use of mix parameters, the two-phase model will give more accurate predictions when there is kinematic disequilibrium. In order to calculate these equations, the model will discretize them using a hybrid between the flux-vector splitting and flux-difference splitting. The reason that the model uses a hybrid of these two discretization techniques is to be able to calculate the contact discontinuities with a high resolution while still being able to run the simulation quickly.

Two-fluid model

The two-phase model is basically two sets of equations for the liquid and the gas. First equations are for the mass balance. These equations describe the changes in the mass for the liquid and the gas. The model is based on a one dimensional case where the fraction, velocity and density are equal all over the pipe cross-section for a given position at a given time. All of the equations are based on iso-thermal conditions. Since the volume of the discrete units used in the simulation are all of equal size, the volume has been divide out of the equation, and the unit for the equations end up being $kg/(m^3s)$ for the SI units.

$$\partial_t(\varepsilon_L\rho_L) + \partial_x(\varepsilon_L\rho_L u_L) = \Gamma_L \quad (20)$$

$$\partial_t(\varepsilon_G\rho_G) + \partial_x(\varepsilon_G\rho_G u_G) = \Gamma_G \quad (21)$$

As previously the subscripts L and G refer to liquid and gas respectively. ε is the fraction of the fluid, ρ is the density of the fluid, u is the velocity of the fluid and Γ is the mass transfer between the two phases. The first expression in the equation is the change of fraction and density of the fluid in time. Since neither the gas nor the liquid is incompressible the density will change over time and the flow is not homogeneous and hence the fraction of the fluid will change over time. The second expression in the equations refers to the flux of mass. Together these expressions represent the mass transfer between the two phases.

Furthermore there must be two equations for the momentum of the two fluids. The momentum also uses a fixed volume, and hence the volume is taken out of the equation and the resulting unit for the equations is force per volume or N/m^3 in SI units.

$$\partial_t(\varepsilon_L\rho_L u_L) + \partial_x(\varepsilon_L\rho_L u_L^2) + \varepsilon_L\partial_x p + \tau_L = M_L^D + q_L \quad (22)$$

$$\partial_t(\varepsilon_G \rho_G u_G) + \partial_x(\varepsilon_G \rho_G u_G^2) + \varepsilon_G \partial_x p + \tau_G = M_G^D + q_G \quad (23)$$

Nomenclature is similar to the one used in the previous equation with the new additions representing: p is the common pressure for the fluid, τ is a differential term that is mathematically relevant since it affects the well-posed system, M^D is the interfacial drag between the fluids and since there are only two fluids in the system the following must be true $M_L^D = -M_G^D$ and the q is a source term that represents the frictional forces and gravitational forces exhibited on the fluid. The first term in the equation is an expression for the change in momentum over time, the second term is an expression for the change in kinetic energy along the x-axis, the third term in the equation represent the change in common pressure along the x-axis and τ is just the mathematical term. These three equations are then equal to the interfacial drag between the phases and the source/sink we get from frictional and gravitational forces.

Assumptions and simplifications for the model

In order to solve the two-phase equations some assumption need to be made since several of the parameters are unknown. The unknown parameters for the equation set are the fraction ε_G and ε_L, density ρ_G and ρ_L, velocity u_G and u_L and the common pressure p. Additionally the system is very difficult to solve in its current state for several reasons: The system is not in conservation law form because of the pressure term, the source term associated with the inter-phase drag are stiff acting on a very short time scale and finally the eigenstructure of the Jacobian matrix for the flux function is unknown. The system might also fail to be hyperbolic which means that there can be oscillations in the numerical solution.

To counter some of these problems a drift-flux model approach will be used where equations (22) and (23) are added together to create a mixture model for the momentum. This means that the interfacial drag is removed and the missing information in the equation sets are replaced by empirical slip, equations which will give the relation between the velocities of the gas and liquid. The equation will also be on a conservative form. Since the drift-flux model can be show to be hyperbolic on reasonable range of values, there will not be problems with oscillations in the numerical solutions. For the drift-flux case equations (22) and (23) are rewritten as:

$$\partial_t(\varepsilon_L \rho_L u_L + \varepsilon_G \rho_G u_G) + \partial_x(\varepsilon_L \rho_L u_L^2 + \varepsilon_G \rho_G u_G^2 + p) = -q \quad (24)$$

Equations (20) and (21) remain the same.

In addition to rewriting the formula as a drift-flux formula, some other assumptions need to be added in order for the system to be solvable. First it is assumed that there is no transfer of mass between the phases. In mathematical terms it means:

$$\Gamma_G = \Gamma_L = 0 \quad (25)$$

Furthermore it is assumed that the velocity of the gas can be written on the analytical slip form:

$$u_g = KU_{Mix} + S \quad (26)$$

K and S in this equation are parameter based on the flow parameters and flow regime.

Since the model is based on iso-thermal conditions the density of the liquid will only depend on the pressure, it is therefore assumed that the density at a given pressure can be expressed as:

$$\rho_L = \rho_{L,ref} + \frac{p - p_{ref}}{a_L^2} \quad (27)$$

In the equation the ref subscript means a reference value for the pressure and density, for example the density of the liquid at reference pressure of 1 bar. a_L represents the velocity of sound in the liquid phase.

The density of the gas is assumed to only be related to the pressure and the velocity of sound in the gas a_G.

$$\rho_G = \frac{P}{a_G^2} \quad (28)$$

Finally the source/sink term for forces are split up into two different parameters, one is representing the friction (F_W) and the other representing the gravity (F_G). The sum of these two values is equal to the q. The gravitational force can be written:

$$F_G = g(\varepsilon_L \rho_L + \varepsilon_G \rho_G) \sin\theta \quad (29)$$

For the frictional force the viscosity is assumed to be an average weighted base on the fraction, and the expression for the force is then:

$$F_W = \frac{32 U_{Mix} \mu_{Mix}}{d^2} \quad (30)$$

To complete the calculation an expression for the velocity of sound in the mixture needs to be created, the following correlation is being used in the model:

$$\omega^2 = \frac{p}{\varepsilon_L \rho_L (1 - K\varepsilon_G)} \quad (31)$$

Where K is a distribution parameter, and is generally a flow dependent constant.

The eigenvalues are then given by:

$$\lambda_1 = u_L - \omega \quad (32)$$

$$\lambda_2 = u_G \quad (33)$$

$$\lambda_3 = u_L + \omega \quad (34)$$

Where the first and third eigenvalues represent the pressure pulse traveling downstream and upstream and the second eigenvalue is the wave speed of the gas volume wave traveling downstream.

Splitting scheme and numerical schemes

The total system can be written in a conservative vector form:

$$\partial_x w + \partial_t F(w) = G(w) \quad (35)$$

Where the symbols represent the following:

$$w = \begin{pmatrix} \varepsilon_L \rho_L \\ \varepsilon_G \rho_G \\ \varepsilon_L \rho_L u_L + \varepsilon_G \rho_G u_G \end{pmatrix} \quad (36)$$

$$F(w) = \begin{pmatrix} \varepsilon_L \rho_L u_L \\ \varepsilon_G \rho_G u_G \\ \varepsilon_L \rho_L u_L^2 + \varepsilon_G \rho_G u_G^2 + p \end{pmatrix} \quad (37)$$

$$G(w) = \begin{pmatrix} 0 \\ 0 \\ -q \end{pmatrix} \quad (38)$$

This can be simplified slightly by assuming that:

$$w_1 = \varepsilon_L \rho_L \quad (39)$$

$$w_2 = \varepsilon_G \rho_G \quad (40)$$

$$w_1 = \varepsilon_L \rho_L u_L + \varepsilon_G \rho_G u_G \quad (41)$$

Then the entire system will have this form:

$$\partial_t \begin{pmatrix} w_1 \\ w_2 \\ w_3 \end{pmatrix} + \partial_x \begin{pmatrix} u_L w_1 \\ u_G w_2 \\ u_L^2 w_1 + u_G^2 w_2 + p(w_1, w_2) \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ -q \end{pmatrix} \quad (42)$$

The flux in the equation (42) is the split up to treat the convection and the pressure separately in the discretization, the convective flux is also split up in to liquid and gas flux, this yields the following set of equations:

$$F = F_{c,L} + F_{c,G} + F_p \quad (43)$$

Where:

$$F_{c,L} = \varepsilon_L \rho_L u_L \begin{pmatrix} 1 \\ 0 \\ u_L \end{pmatrix} \quad (44)$$

$$F_{c,G} = \varepsilon_G \rho_G u_G \begin{pmatrix} 0 \\ 1 \\ u_G \end{pmatrix} \quad (45)$$

$$F_p = \begin{pmatrix} 0 \\ 0 \\ p \end{pmatrix} \quad (46)$$

To avoid excessive dissipation at the contact discontinuities associated with the FVS type scheme, a scheme that eliminated this dissipation is needed. Therefore the scheme will draw from both the FVS and the FDS schemes. The fundamental idea is to identify a "convective speed" and implement upwind principles in the discretization of the convective terms. Based on the slipping done in the previous equations a natural choice for the interface convective speed is the mass fluxes $(\varepsilon_L \rho_L u_L)_{j+1/2}$ for the liquid and $(\varepsilon_G \rho_G u_G)_{j+1/2}$ for the gas. Numerically the flux at the interface between j and j+1 can then be written:

$$F_{j+1/2}(w_L, w_G) =$$

$$\frac{1}{2}\left[(\varepsilon_L\rho_L u_L)_{j+1/2}(\Phi_{L,L} + \Phi_{L,R}) - \left|(\varepsilon_L\rho_L u_L)_{j+1/2}\right|(\Phi_{L,R} - \Phi_{L,L})\right] +$$

$$\frac{1}{2}\left[(\varepsilon_G\rho_G u_G)_{j+1/2}(\Phi_{G,L} + \Phi_{G,R}) - \left|(\varepsilon_G\rho_G u_G)_{j+1/2}\right|(\Phi_{G,R} - \Phi_{G,L})\right] +$$

$$\left(F_p\right)_{j+1/2} \quad (47)$$

Where:

$$\Phi_L = (1,0,u_L)^T \quad (48)$$

$$\Phi_G = (0,1,u_G)^T \quad (49)$$

$$F_p = (0,0,p)^T \quad (50)$$

The following Van-Leer FV/DS and AUSM type scheme is used for the two-phase flow, with equal discretization for both the fluids. They are based on the work done by Wada and Liou[4]. Here the R and L subscript represent right and left respectively.

Van-Leer type:

$$(\varepsilon\rho u)_{j+1/2}^{Van-Leer} = V^+\left(u_L, c_{j+1/2}\right)\varepsilon_L\rho_L + V^-\left(u_R, c_{j+1/2}\right)\varepsilon_R\rho_R \quad (51)$$

The AUSM type:

$$(\varepsilon\rho u)_{j+1/2}^{AUSM} = \frac{1}{2}\left[u_{j+1/2}(\varepsilon_L\rho_L + \varepsilon_R\rho_R) - \left|u_{j+1/2}\right|(\varepsilon_R\rho_R - \varepsilon_L\rho_L)\right] \quad (52)$$

Where:

$$u_{j+1/2} = u_L^+ + u_R^- = V^+\left(u_L, c_{j+1/2}\right) + V^-\left(u_R, c_{j+1/2}\right) \quad (53)$$

Sound velocity is there common which means that the AUSM M-splitting and U-splitting will coincide. Formula for the velocity splitting is here defined as:

$$V^\pm(u,c) = \begin{cases} \pm\frac{1}{4c}(u \pm c)^2 & if\, |u| \leq c \\ \frac{1}{2}(u \pm |u|) & otherwise \end{cases} \quad (54)$$

The consistency condition is here satisfied for the velocity splitting function since their sum is equal to u.

An FVS type discretization term is used for the pressure term, which results in the following equation:

$$p_{j+1/2} = P^+(u_L, c_{j+1/2})p_L + P^-(u_R, c_{j+1/2})p_R \quad (55)$$

In this formula u represents the mixed fluid velocity, assumed here to be the sum of the superficial velocities since the fluxes are assumed to be subsonic, and the splitting formulas for the pressure are given by:

$$P^\pm(u, c) = V^\pm(u, c) \cdot \begin{cases} \frac{1}{c}\left(\pm 2 - \frac{u}{c}\right) & if\,|u| \leq c \\ \frac{1}{u} & otherwise \end{cases} \quad (56)$$

Sound velocities used in the previous formulas are related to the mixture, in the model the following approximation is used to determine the sound velocity.

$$c(\varepsilon_G) = \begin{cases} a_L & if\,\varepsilon_G < \kappa \\ \omega & if\,\epsilon < \varepsilon_G < 1 - \kappa \\ a_G & if\,\varepsilon_G > 1 - \kappa \end{cases} \quad (57)$$

The value for ω is give by formula (31) and the vale for κ is just a small value to ensure that there is a smooth transition from two-phase to single-phase flow.

The common sound speed in the interface between j and j+1 is the average between the sound speed in j and j+1, however in this model c=max(c_L, c_R).

For the FVS type scheme the discretization of the pressure term remain the same, however the convective term are being discretized in the following way:

$$F_{j+1/2}^{FVS}(w_L, w_R) = (\varepsilon_L\rho_L)_L \Psi_{L,L}^+ + (\varepsilon_L\rho_L)_R \Psi_{L,R}^- + (\varepsilon_G\rho_G)_L \Psi_{G,L}^+ + (\varepsilon_G\rho_G)_R \Psi_{L,R}^- + (F_p)_{j+1/2} \quad (58)$$

Where:

$$F_p = (0,0,p)^T \quad (59)$$

$$\Psi^+_{L,L} = \Psi^+_L(u_{L,L}, c_{j+1/2}) \quad (60)$$

$$\Psi^-_{L,R} = \Psi^+_L(u_{L,R}, c_{j+1/2}) \quad (61)$$

$$\Psi^+_{G,L} = \Psi^+_G(u_{G,L}, c_{j+1/2}) \quad (62)$$

$$\Psi^-_{G,R} = \Psi^+_G(u_{G,R}, c_{j+1/2}) \quad (63)$$

$$\Psi^+_L(u, c) = V^+(u, c)\begin{pmatrix} 1 \\ 0 \\ u \end{pmatrix} \quad (64)$$

$$\Psi^-_L(u, c) = V^-(u, c)\begin{pmatrix} 1 \\ 0 \\ u \end{pmatrix} \quad (65)$$

$$\Psi^+_G(u, c) = V^+(u, c)\begin{pmatrix} 0 \\ 1 \\ u \end{pmatrix} \quad (66)$$

$$\Psi^-_G(u, c) = V^-(u, c)\begin{pmatrix} 0 \\ 1 \\ u \end{pmatrix} \quad (67)$$

The velocity splitting remains the same as in equation (54). The FVS type scheme has similar mass conservation equations as the Van-Leer FV/DS, however the mixed momentum equation have a different discretization. Since:

$$F^{FVS}_{j+\frac{1}{2}}(w, w) = F^-(w) + F^+(w) \quad (68)$$

The scheme is really a flux vector splitting scheme.

About the dissipative mechanism of the schemes, the AUSM mass flux can be rewritten to the viscous form:

$$(\varepsilon_k \rho_k u_k)^{AUSM}_{j+1/2} = u_{k,j+1/2} \frac{(\varepsilon_k \rho_k)_L + (\varepsilon_k \rho_k)_R}{2} - \frac{1}{2} d^{AUSM}_{k,j+1/2} \quad (69)$$

$$d^{AUSM}_{k,j+1/2} = |u_{k,j+1/2}|((\varepsilon_k \rho_k)_R - (\varepsilon_k \rho_k)_L) \quad (70)$$

The k is here used to represent either of the fluids. This coincides with the mass flux for the FVS and Van-Leer schemes. The FVS and Van-Leer mass flux can be written as follows:

$$(\varepsilon_k \rho_k u_k)_{j+1/2}^{FVS} = \frac{(u_k \varepsilon_k \rho_k)_L + (u_k \varepsilon_k \rho_k)_R}{2} - \frac{1}{2} d_{k,j+1/2}^{FVS} \quad (71)$$

$$d_{k,j+1/2}^{FVS} = |V|(u_{k,R}, c_{j+\frac{1}{2}}(\varepsilon_k \rho_k)_R - |V|(u_{k,L}, c_{j+\frac{1}{2}}(\varepsilon_k \rho_k)_L) \quad (72)$$

In the equation the absolute value of the velocity splitting function represent the positive one minus the negative one. The mass flux of the FVS is given from:

$$(\varepsilon_k \rho_k u_k)_{j+1/2} = (a_k)_L V_{k,L}^+ + (a_k)_R V_{k,R}^- \quad (73)$$

Where:

$$a = \varepsilon \rho \quad (74)$$

$$V_{k,L}^+ = V^+(u_{k,L}, c_{j+1/2}) \quad (75)$$

$$V_{k,R}^- = V^-(u_{k,R}, c_{j+1/2}) \quad (76)$$

From the fact that the velocity splitting functions sum to u, it follows that:

$$(\varepsilon_k \rho_k u_k)_{j+1/2} = (a_k)_L (u_{k,L} - V_{k,L}^-) + (a_k)_R (u_{k,R} - V_{k,R}^+) \quad (77)$$

By adding the equation (71) together with the equation (75) it becomes:

$$(\varepsilon_k \rho_k u_k)_{j+1/2} =$$

$$\frac{(\varepsilon_k u_k)_L + (\varepsilon_k u_k)_R}{2} - \frac{1}{2}\left[(V_{k,R}^+ - V_{k,R}^-)(a_k)_R - (V_{k,L}^+ - V_{k,L}^-)(a_k)_L\right] \quad (78)$$

Considering numerical dissipation at steady and moving contact discontinuities, firstly it is assumed that there is a moving contact discontinuity where:

$$p_L = p_R = p \quad (79)$$

$$u_{G,L} = u_{G,R} = u_G \quad (80)$$

$$u_{L,L} = u_{L,R} = u_L \quad (81)$$

$$\varepsilon_{G,L} \neq \varepsilon_{G,R} \quad (82)$$

From equation (69) the mass flux for AUSM will become:

$$(\varepsilon_k \rho_k u_k)^{AUSM}_{j+1/2} = u_k \rho_k \frac{\varepsilon_{k,R} + \varepsilon_{k,L}}{2} - |u_k| \rho_k \frac{\varepsilon_{k,R} - \varepsilon_{k,L}}{2} \quad (83)$$

Where:

$$
\begin{array}{ll}
u_k \rho_k \varepsilon_{k,L} & if \ u_k > 0 \\
u_k \rho_k \varepsilon_{k,R} & otherwise
\end{array} \quad (84)
$$

$$u_{k,j+1/2} = V^+(u_k, c_{j+1/2}) + V^-(u_k, c_{j+1/2}) = u_k \quad (85)$$

For the AUSM scheme the steady contact discontinuity, where $u_k = 0$, has a vanishing numerical dissipation and the value of expressing (83) is zero.

Considering the FVS and Van-Leer schemes the absolute value of the velocity splitting function is:

$$|V|(u,c) = \begin{cases} |u| & if \ |u| > c \\ \frac{1}{2}\left[\frac{u^2}{c} + c\right] & otherwise \end{cases} \quad (86)$$

The mass flux for the FVS and Van-Leer scheme in the sub sonic range is then given by:

$$(\varepsilon_k \rho_k u_k)^{FVS}_{j+1/2} = u_k \rho_k \frac{\varepsilon_{k,R} + \varepsilon_{k,L}}{2} - \frac{1}{2}\left(\frac{u_k^2}{c_{j+1/2}} + c_{j+1/2}\right) \rho_k \frac{\varepsilon_{k,R} - \varepsilon_{k,L}}{2} \quad (87)$$

This will for a stationary contact discontinuity not be equal to zero but will become:

$$(\varepsilon_k \rho_k u_k)^{FVS}_{j+1/2} = -c_{j+1/2} \rho_k \frac{\varepsilon_{k,R} - \varepsilon_{k,L}}{4} \quad (88)$$

In the AUSMD and AUSMV type schemes the velocity splitting function is being replace by a more general pair of equations:

$$\tilde{V}^\pm(u,c,\chi) = \begin{cases} \chi V^\pm(u,c) + (1-\chi)\frac{u \pm |u|}{2} \\ \frac{1}{2}(u \pm |u|) \qquad otherwise \end{cases} \quad for |u| \leq c \quad (89)$$

Where the new velocity splitting function is a convex combination of the $V^{\pm}$ and $(u\pm|u|)/2$, and the sum of the positive and negative velocity splitting function is still equal to u. The new velocity splitting function is inserted into the FVS and Van-Leer FV/FD schemes, the new schemes gotten from this is denoted AUSMD and AUSMV. The variable χ is selected to get a better value for the numerical fluxes with respect to accurate approximations of steady and moving contact discontinuities. The AUSMV and AUSMD mass flux is now given by the following equation:

$$(\varepsilon\rho u)_{j+1/2} = (\varepsilon\rho)_L \tilde{V}_L^+ + (\varepsilon\rho)_R \tilde{V}_R^- \quad (90)$$

$$\tilde{V}_L^+ = \tilde{V}^+(u_L, c_{j+1/2}, \chi_L) \quad (91)$$

$$\tilde{V}_R^- = \tilde{V}^-(u_R, c_{j+1/2}, \chi_R) \quad (92)$$

The viscous term in the equation will become:

$$d_{j+1/2}^{AUSMV} = |\tilde{V}|(u_R, c_{j+1/2}, \chi_R)(\varepsilon\rho)_R - |\tilde{V}|(u_L, c_{j+1/2}, \chi_L)(\varepsilon\rho)_L \quad (93)$$

Where:

$$|\tilde{V}|(u, c, \chi) = \tilde{V}^+(u, c, \chi) - \tilde{V}^-(u, c, \chi) = \chi|V|(u, c) + (1-\chi)|u| \quad (94)$$

The purpose is to choose values for χ, such that the mass flux of the FVS and Van-Leer scheme takes the same form as the mass flux AUSM at stationary and moving gas fraction contact discontinuity.

For a stationary contact discontinuity where u=0, the formula become:

$$d_{j+1/2}^{AUSMV} = |\tilde{V}|\left(0, c_{j+\frac{1}{2}}, \chi_R\right)(\varepsilon\rho)_R - |\tilde{V}|\left(0, c_{j+\frac{1}{2}}, \chi_L\right)(\varepsilon\rho)_L =$$

$$|V|(0, c_{j+\frac{1}{2}})\rho(p)[\varepsilon_R \chi_R - \varepsilon_L \chi_L] \quad (95)$$

Since the goal is to not gain any numerical dissipation in the mass fluxes at the steady contact discontinuity values for χ will be selected so that:

$$\chi_R \varepsilon_R - \chi_L \varepsilon_L = 0 \quad (96)$$

For a moving contact discontinuity the equation (93) will take the following form:

$$d_{j+1/2}^{AUSMV} = |\tilde{V}|(u_R, c_{j+1/2}, \chi_R)(\varepsilon\rho)_R - |\tilde{V}|(u_L, c_{j+1/2}, \chi_L)(\varepsilon\rho)_L \quad =$$

$$|u|\rho(p)[\varepsilon_R - \varepsilon_L] \quad (97)$$

In light of formulas (71), (93) and (97) the mass flux will become:

$$(\varepsilon\rho u)_{j+1/2}^{AUSMV} = u\rho(p)\frac{\varepsilon_R+\varepsilon_L}{2} - \frac{1}{2}|u|\rho(p)[\varepsilon_R - \varepsilon_L] \quad (98)$$

Several values can be chosen for χ, however the easiest choice is to set χ=ε.

Simulation program

Main routine in the program

The routine starts up by loading the data from the PARAC1_FL.DEF data file. This data is then assigned to its respective parameters. Geometric data for the pipe, reference values for pressure and density and the sound velocity in the fluids is put in after this, this does not load from a data file but is put in straight into the routine. The grid size is then determined based on the length of the pipe and the number of grid cells. Initial data for U_o, C1, C2, CC1 and CC2 is then assigned, the last four of these parameters are drift-flux parameters and C1 and C2 correspond to K and S in equation (24) respectively. Then there is created zero vectors for these parameters, U_o being the conservative variables requires a 3 by n zero matrix. Subroutine init_funcC1_FL is then called in order to calculate the drift-flux parameters and the matrix for the conservative variables. These calculations are done based on the position, sound velocity, reference pressure and density. Subplots are the created for the conservative variables against the position in the pipe. Then the program creates zero vectors for the initial pressure, fluid density, fluid velocity and fluid fraction, or p_0, ρ_{L0}, ρ_{G0}, u_{L0}, u_{G0}, ε_{L0} and ε_{G0} respectively. PrimitivC1_FL is the called in order to calculate these parameters, the basis for the calculations are the conservative variables, fluid sound velocity, reference pressure and density and the drift-flux parameters. Subplots for the pressure, fluid velocity and fluid fraction against the position in the pipe are created. Size of the time steps are calculated based on the total time and the number of time steps. Zero vectors for mass flow of the liquids are created, and the massflow_in subroutine is used to calculate the values for these vectors. Subplots are created for these mass flows against the time. Likewise zero vectors for the mass flow out are created, and the values of these vectors are calculated from the subroutine massflow_out. Subplots are created for the mass flow out against time for the fluids. Zero

vectors of the conservative variables and the physical fluxes inflow and outflow are created, note that the conservative variables here are called v and not u like in the rest of the program. The primitive variables at the inflow are set equal to the values in grid one and the outflow values are set equal to the values in grid n. The program CvarPflux is then used to calculate the inflow and outflow values for the conservative variables and the physical fluxes, based on the primitive variables at the inflow and outflow. All these calculation are done to get the correct boundary treatment. The solution procedure for the entire pipe starts, zero vectors and matrixes are set for the primitive variables, conservative variables and the physical fluxes. In addition zero vectors for the ω and λ variables are created, these parameters are used to calculate the eigenvectors. Two zero matrixes for a pre value for the conservative variables and the physical fluxes are also created, the pre values are just going to be based on some initial data and calculations as input in the sol_stian subroutine. The pre value for the conservative variables are set equal to the conservative variables calculated initially U_o. Using the FfluxC1_FL subroutine the pre values of the physical fluxes are calculated together with the values for the primitive variables. FfluxC1_FL uses pre values of the conservative variable, the fluid sound velocity, reference pressure and density and the drift-flux parameters to determine these values. Global time steps for the calculations are set based on time and number of steps. Zero vectors for the liquid rate, gas rate and relative pressures are created, for each of these four vectors are created representing different positions in the pipe. The values of these vectors are then calculated based on the values gotten from the previous FfluxC1_FL calculation. A for loop is then created, this loop will do the calculation of all the variables for all the given time steps. The subroutine sol_stian is used to determine the conservative variables and physical fluxes at the inflow and outflow, conservative variables over the entire pipe and the drift-flux parameters. All these calculations are based on the pre value of the conservative variables, fluid sound velocity, reference pressure and density, initial drift-flux parameters, boundary values for conservative variables and physical fluxes, pipe geometry, initial fluid parameters, position and time. Since the default case is a case where the pipe is close at the top the program checks that the mass of the fluids are conserved. Form the conservative variables, fluid sound velocity, reference pressure and density, drift-flux parameters and time the primitive variables over the entire pipe is calculated using the primitivC1_FL subroutine. Then the sound velocity for the fluid mixture is calculated together with the eigenvectors. Gas rate, liquid rate and the relative pressure is calculated for the four different points in the pipe previously determined, based on the primitive variables. Graphs

are printed to display the calculated data and the initial data is updated for the next time step calculation.

Subroutines in the program

CvarPflux

This subroutine calculates the conservative fluxes and the conservative variables based on the fluid density, fraction, velocity and pressure. The calculations are done based of equation (36) and (37).

FfluxC1_FL

The fluxes are determined by this program. The input data are used together with sub routine "primitivC1_FL" to calculate the three vectors for the flux, which again is based on equation (37).

init_funcC1_FL

From this subroutine the conservative variables and drift flux parameters are calculated. The routine has built in several ways to determine these parameters, however they are mostly based on equation (27), (28) and (36). The ways the slip parameters are calculated are based on different empirical correlations.

massflow_in

This subroutine determine the fluids flow into the pipe, there are several preset ways the fluids can flow in. The flow into the pipe can vary over time or based on other parameters to fit the case that is going to be simulated.

massflow_out

This subroutine is the opposite of the previous subroutine, determining how and when the fluids will flow out of the pipe.

primitivC1_FL

Form this subroutine the fluids density, velocity, fraction and the pressure is calculated based on the conservative variables, sound velocities, reference density, reference pressure, and drift flux parameters. The subroutine first calculate the pressure based on a thermo dynamical step and when the pressure is know the densities, fractions and velocities are calculated. The

thermo dynamical step is based on equations (27), (28) and (36) with some algebraic operations to be able to solve for P.

<u>sol_stian</u>

This subroutine is used to calculate the conservative variables and the physical fluxes in and out and to update the drift-flux parameters. The calculations are based on all the primitive variables, the initial assumed conservative variables, physical fluxes and drift flux parameters. Initially zero vectors are created for all the parameters that are going to be calculated during the routine. The conservative variables and physical fluxes at node one is set equal to the in values give by the main program and the values at n set equal to the out values. Some basic derivation functions and some vectors used for calculations are defined. Then the primitive variables are calculated from the primitivC1_FL subroutine, and vectors for the conservative variables are defined and the physical fluxes are calculated from the FflucC1_FL subroutine. Drift-flux parameters are then calculated based on the selected scheme. The calculations for the fluid velocity, fractions and densities will use a second order slope. This means that the derivative of the variable for grid j is calculated to find the value of the variable at the interface between the two grids. For example for the density:

$$\rho_{L,L} = \rho_{L,j} + 0.5 \frac{d\rho_j}{dx} dx \quad (99)$$

$$\rho_{L,R} = \rho_{L,j+1} - 0.5 \frac{d\rho_{j+1}}{dx} dx \quad (100)$$

The reason that this calculation is done is to increase the accuracy of the calculation without increasing the number of grids. This means that the amount of CPU power used is lowered. The calculation will set the derivative equal to zero if there are discontinuities, and in these cases the parameter is equal to the value in the middle of the grid. After these calculations are done, the program will have left and right values for the fluid velocity, fraction and density for both the fluids. Based on these values the sound velocity for the left and right is calculated. The greatest of these values are selected as the sound velocity for the further calculations. Then the values for χ left and right is calculated based on the selected scheme and from these values the program is able to calculate the splitting velocities. Formulas for the splitting velocities are given at the end of the program. Temporary values for the fluxes are calculated. The friction term is calculated from the SourceFric subroutine. Together with the values previously determined, the physical fluxes for the mass balance equations are

calculated and put into the corresponding matrix. Using the UpdateP the pressures are calculated for all the grids. Then using the same second order slop as used previously, but with slightly different scheme for the derivation, the pressure left and right can be determined. The pressure splitting values are determined and then values for the mixture momentum and energy equation can then be calculated. Using this matrix for the fluxes as basis the calculations are then repeated to get a solution that converges closer to the actual solution. Using the massflow_in and massflow_out subroutines the boundary behavior is updated from CvarPflux. The boundary behavior of the fluxes, the matrix for the fluxes and drift-flux parameters are then exported to the main program.

SourceFric

This subroutine is used in the sol_stian in order to calculate the frictional and gravitational pressure drop in the pipe, based on the primitive variables, position, time and geometric data for the pipe. The frictional pressure drop is based on formula (30) and the gravitational pressure drop is calculated from formula (29). The primitive variables for the mixture are the fractional weighted average of the fluids.

UpdateP

This subroutine calculates the pressure, fluid fraction and density based on the knowledge of conservative parameters, sound velocity, reference pressure, liquid reference density and the drift-flux parameters.

Plotting routines in the program

3D plotting routines

The 3D plotting routines all have the same basis, but display different things. Information is loaded from the data files in the solution folder, and the information is put into the designated matrixes. Then a 3D plot is created where the x-axis is the length of the pipe, y axis is time and z-axis is a varying parameter base on which of the routines are use. In the routines the number of grids M used in the calculation, total time NT and size of time steps must be defined. The y parameter corresponds to:

Plot3D_alpG plots the gas fraction ε_G.

Plot3D_P plots the pressure p in bar.

Plot3D_Vg_Sup plots the velocity of the gas in m/s.

Plot3D_VgVl plots the gas and liquid velocity in m/s.

Plot3D_VgVl_Sup plots the gas and liquid velocity in m/s and the gas fraction ε_G.

Plot3D_Vl_Sup plots the velocity of the liquid in m/s.

2D plotting routines

The 2D plotting routines makes a simple x and y plot of some chosen parameters, the parameters are based on which program is selected. Like for the 3D plots the routine load the data from the data files and put the information into the designated matrixes. There are two types of plotting routines for the 2D, the one which plots the initial conditions at either the top or bottom of the pipe and the one that plots for a set time step. The routines that plot the initial conditions don't require any input data except the data files. The routines that plot for a given time step however, needs input about the number of grids used in the simulation and the number corresponding to the desired time step. Initial condition plotting routines are to following:

Tegn_alphG0_x plots the initial values for ε_G along the y-axis and distance along the pipe in meter on the x-axis.

Tegn_p0_x plots the initial values for the pressure in bar along the y-axis and the distance along the pipe in meter on the x-axis.

Tegn_p_t plots the values of the pressure in bar at top and bottom on the y-axis and time in second on the y-axis.

Tegn_Vg0_x plot the initial value of the gas velocity in m/s along the y-axis and the distance of the pipe in meter along the x-axis.

Tegn_Vg_t plots the gas mass flux in kg/s at the top of the pipe on the y-axis and time in seconds along the x-axis.

Tegn_Vl0_x plot the initial value of the liquid velocity along the y-axis and the distance of the pipe along the x-axis.

Tegn_Vl_t plots the liquid mass flux in kg/s at the top of the pipe on the y-axis and time in seconds along the x-axis.

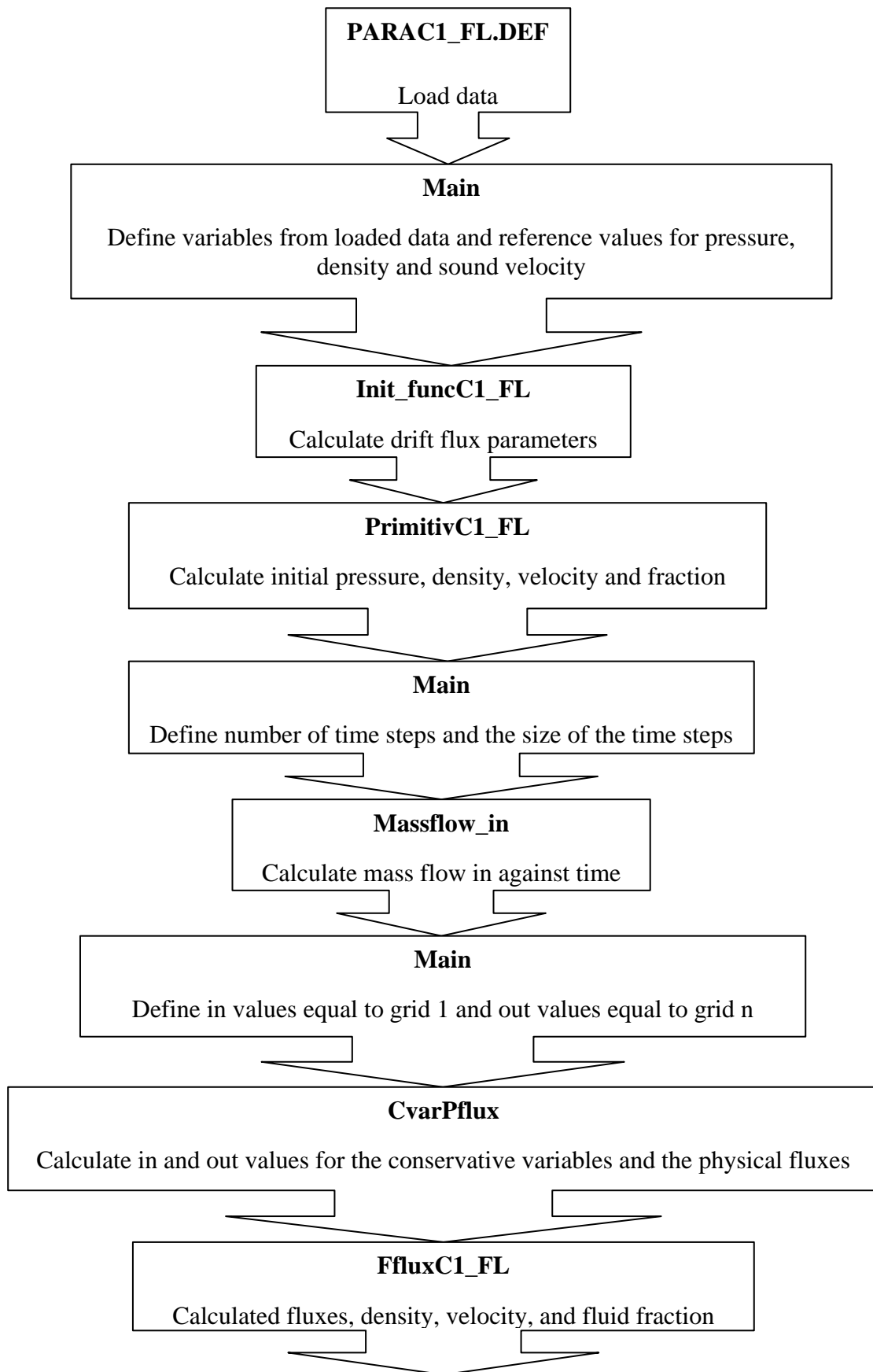Routines requiring additional information other the data files are:

Tegn_alphG_x plot gas fraction against the distance in meter for a given time step.

Tegn_p_x plots pressure in bar against the distant for a give time step.

Tegn_Vg_x plots gas velocity in m/s against the distant for a give time step.

Tegn_Vl_x plots liquid velocity in m/s against the distant for a give time step.
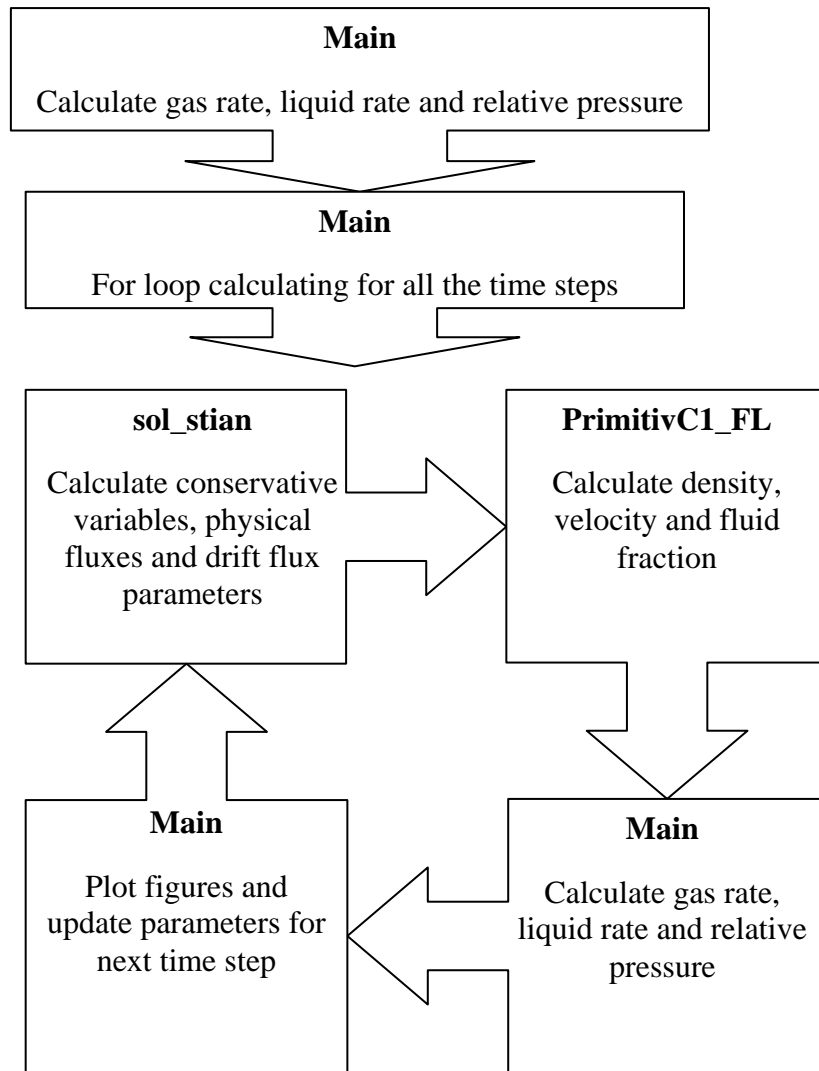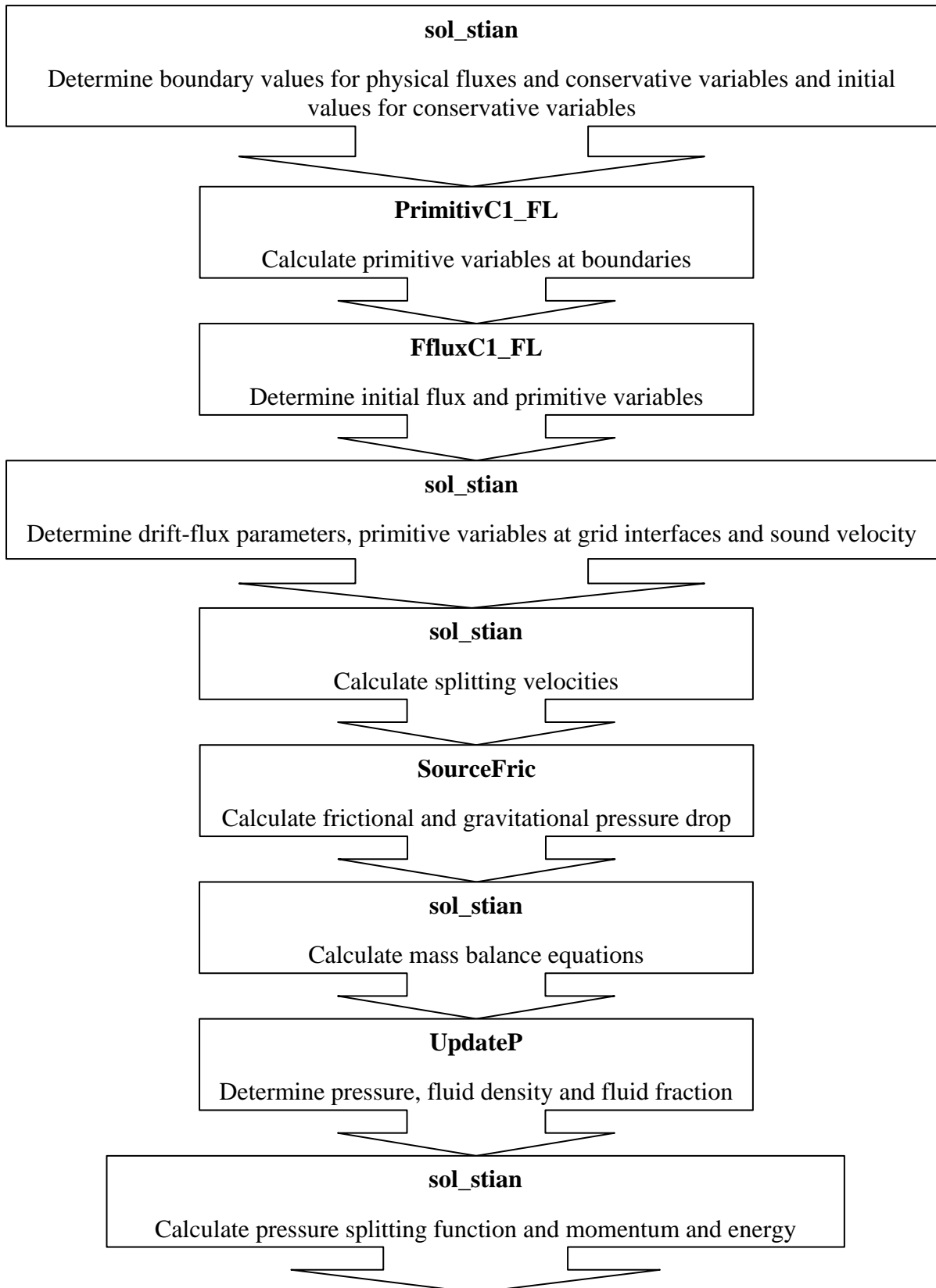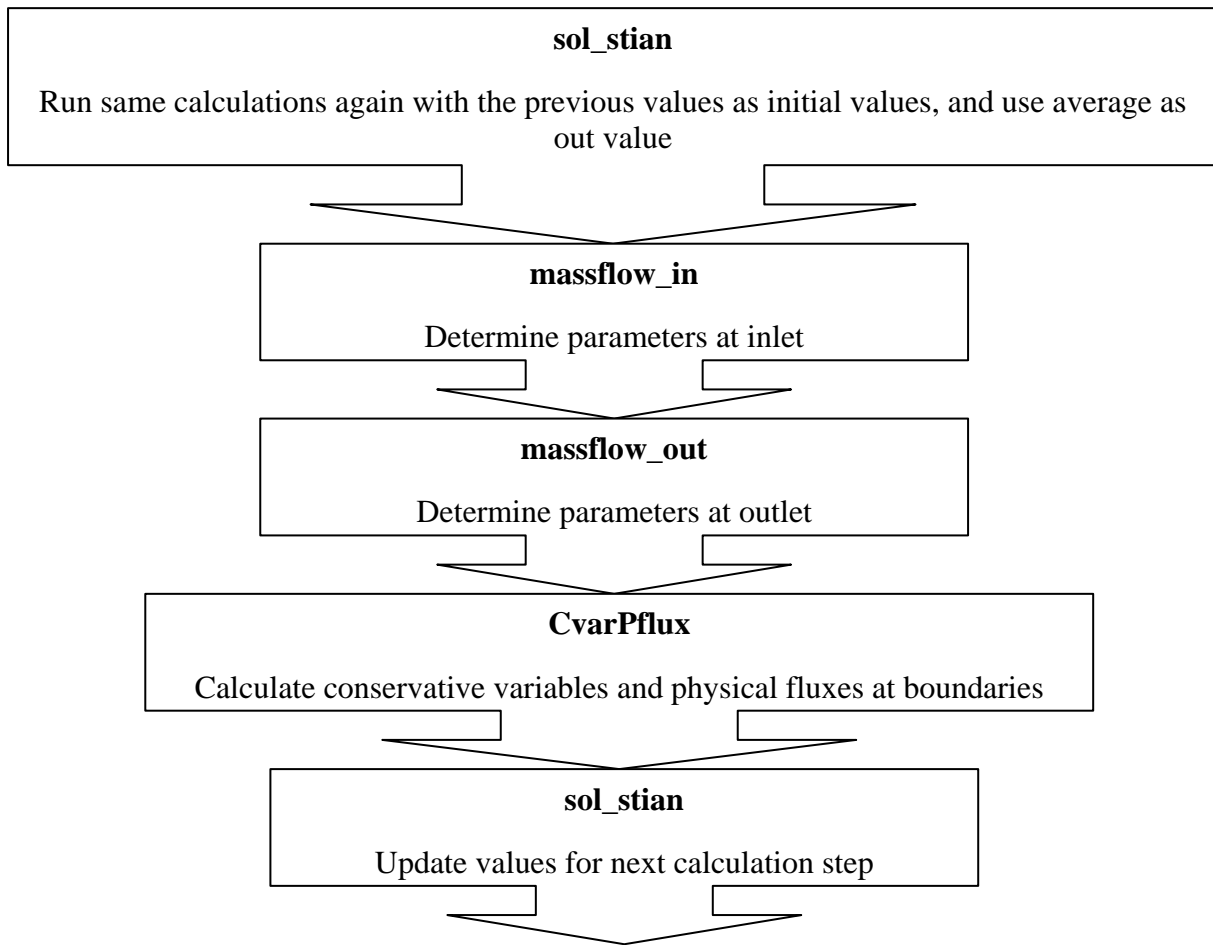
Flowchart showing how the routines work together


```
                    ┌─────────────────────────┐
                    │      PARAC1_FL.DEF       │
                    │                          │
                    │        Load data         │
                    └─────────────────────────┘

  ┌──────────────────────────────────────────────────────────┐
  │                          Main                             │
  │                                                          │
  │   Define variables from loaded data and reference values │
  │      for pressure, density and sound velocity            │
  └──────────────────────────────────────────────────────────┘

              ┌───────────────────────────────────┐
              │          Init_funcC1_FL            │
              │                                    │
              │     Calculate drift flux parameters│
              └───────────────────────────────────┘

        ┌─────────────────────────────────────────────────┐
        │                 PrimitivC1_FL                    │
        │                                                 │
        │  Calculate initial pressure, density, velocity  │
        │                  and fraction                    │
        └─────────────────────────────────────────────────┘

        ┌─────────────────────────────────────────────────┐
        │                      Main                         │
        │                                                  │
        │  Define number of time steps and the size of the │
        │                   time steps                      │
        └─────────────────────────────────────────────────┘

              ┌───────────────────────────────────┐
              │            Massflow_in             │
              │                                    │
              │   Calculate mass flow in against time│
              └───────────────────────────────────┘

        ┌─────────────────────────────────────────────────┐
        │                      Main                         │
        │                                                  │
        │  Define in values equal to grid 1 and out values │
        │               equal to grid n                     │
        └─────────────────────────────────────────────────┘

  ┌──────────────────────────────────────────────────────────┐
  │                        CvarPflux                          │
  │                                                          │
  │  Calculate in and out values for the conservative        │
  │         variables and the physical fluxes                │
  └──────────────────────────────────────────────────────────┘

        ┌─────────────────────────────────────────────────┐
        │                  FfluxC1_FL                       │
        │                                                  │
        │  Calculated fluxes, density, velocity, and fluid │
        │                   fraction                        │
        └─────────────────────────────────────────────────┘
```

Figure 5

sol_stian

sol_stian
Run same calculations again with the previous values as initial values, and use average as out value

massflow_in
Determine parameters at inlet

massflow_out
Determine parameters at outlet

CvarPflux
Calculate conservative variables and physical fluxes at boundaries

sol_stian
Update values for next calculation step

Figure 6

Description on how to use the program

Before the program can be run some parameters need to be defined. In PARAC1_FL.DEF the following values are defined: Total number of grids along the pipe N, the total time for the calculations TIME, maximum sound velocity in the pipe Max, number of time steps the program is going to compute and the viscosity for the fluids. In MainStian the geometric data for the pipe, length L and diameter diam, sound velocity for the fluids, reference pressure and reference density for the liquid is defined. The slip relation and inclination of the pipe is selected in init_funcC1_FL, in this program there are several preset ways to determine the drift-flux parameters and inclination. In this routine the initial distribution of the fluids are also determined as fractions. They way to determine the velocity splitting function and pressure splitting function must be selected in the sol_stian subroutine. The volumetric flows of fluids in and out of the pipe are selected in massflow_in and massflow_out respectively. From this a table can be created to better illustrate where each of the parameters are place if one would like to change them:

Parameter	Location	Letter used in MatLab
Number of grids	PARAC1_FL.DEF	N
Total time	PARAC1_FL.DEF	TIME
Max sound velocity	PARAC1_FL.DEF	Max
Number of time steps	PARAC1_FL.DEF	NSTEP
Viscosity of gas	PARAC1_FL.DEF	muG
Viscosity of liquid	PARAC1_FL.DEF	muL
Length of pipe	MainStian	L
Diameter of pipe	MainStian	Diam
Sound velocity in gas	MainStian	a_G
Sound velocity in liquid	MainStian	a_L
Reference pressure	MainStian	p_{ref}
Reference pressure liquid	MainStian	$\rho_{L,ref}$
Initial fluid distribution	init_funcC1_FL	N/A
Slip relation	init_funcC1_FL	N/A
Pipe inclination	init_funcC1_FL	N/A
Velocity splitting function	sol_stian	N/A
Pressure splitting function	sol_stian	N/A
Volumetric inflow of fluids	massflow_in	N/A
Volumetric outflow of fluids	massflow_out	N/A

Table 1

Once the parameters has been altered to suite the desired case the simulation is begun by opening MainStian and running the program and the pressing enter in the command window.

Simple case to show how the program function

To illustrate how the program function a case will be preformed to demonstrate. In this case the following values have been selected as input parameters:

Parameter	Unit	Value
Number of grids	N/A	50
Total time	s	250
Max sound velocity	m/s	1000
Number of time steps	N/A	250
Viscosity of gas	Pa s	0,000005
Viscosity of liquid	Pa s	0,05

Length of pipe	m	100
Diameter of pipe	m	0,06
Sound velocity in gas	m/s	1000
Sound velocity in liquid	m/s	316,22
Reference pressure	Pa	100000
Reference pressure liquid	kg/m^3	1000

Table 2

Slip parameters in the simulation are determined using the following equations, where K and S represent the slip parameters in equation (26).

$$K = 1{,}2 - 0{,}2 \cdot \varepsilon_G \quad (101)$$

$$S = 2 \cdot (\varepsilon_G + 0{,}2) \cdot (1 - \varepsilon_G) \quad (102)$$

The case is a gas bubble rising in a pipe that is sealed at the end, hence the outflow is set equal to zero, and the inflow is just the initial inflow of the gas. In the init_funcC1_FL it is selected that the initial distribution is oil over the entire pipe, except in grid cell one where there is a gas fraction of 0,99, the pipe has an inclination of 90 degrees and model 2.07 is the chosen slip relation. The velocity splitting functions and pressure splitting functions in sol_stian is model number 2 and 0 respectively. After all the data has been put in to the respective routines, the model yields the following data.

In figure 7 the gas fraction in the pipe is shown for four different times: t=25, t=50, t=100 and t=250, note that the value on the y-axis changes for each figure. This means that despite what it may look like the maximum gas fraction decreases for each step until the gas bubble has reached the top of the pipe. The length of the pipe occupied by the gas is also quite huge at t=100, from the figure at this time step there is gas in at least half of the pipe.
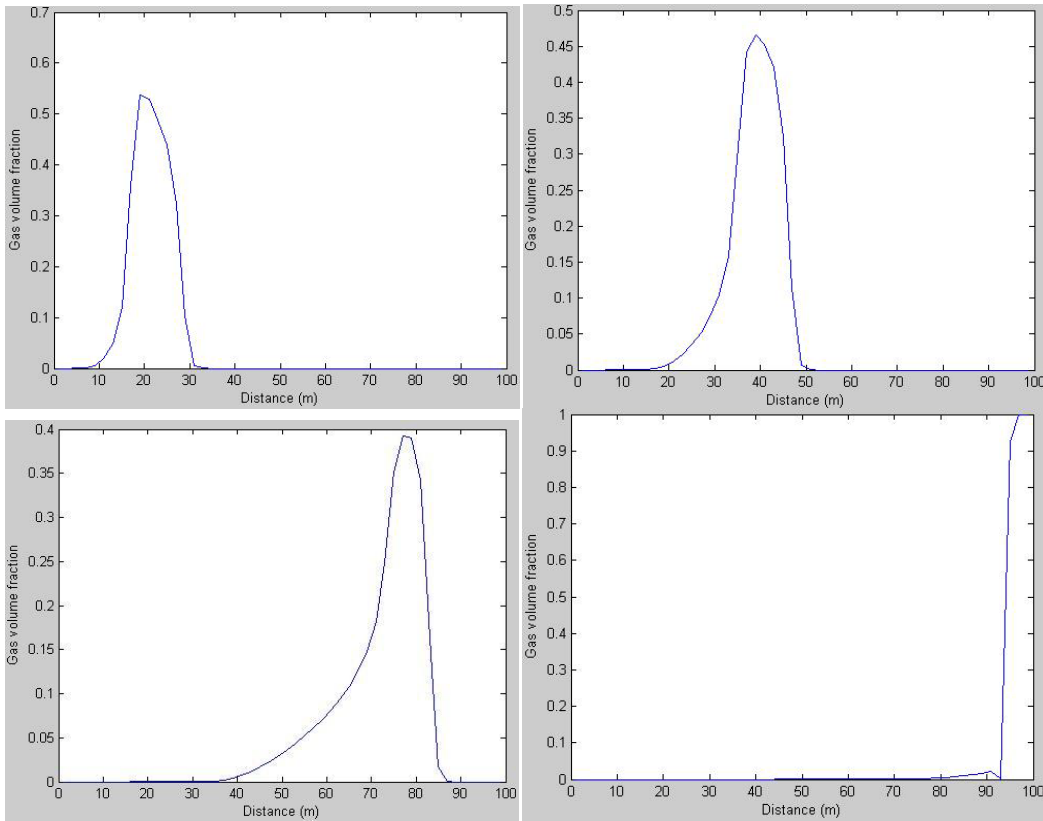
Figure 7
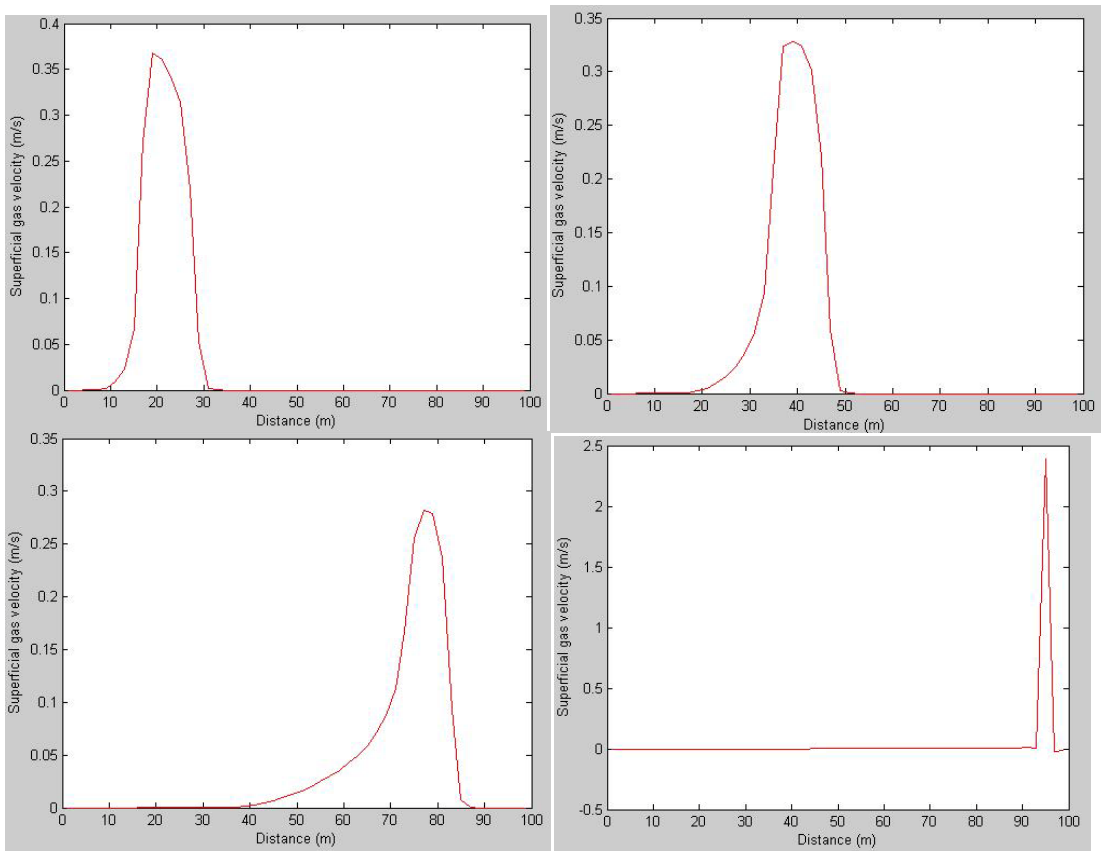
Figure 8

In figure 8 the superficial velocities are plotted against the distance in the pipe for: t=25, t=50, t=100 and t=200. The superficial velocity appears to be declining against time, however at the end it has a sharp increase and almost increasing by a tenfold. The reason that the superficial gas velocity seems to become very high at the end is because the flow in this example strongly depend on the slip relation selected. When the two-phase flow transitions in to single-phase gas getting the correct values for the slip parameters is very demanding, and the slip relations isn't mainly built to handle this transition. However using different and more complex schemes for calculating the slip relation would yield a more physically correct solution. These more complex schemes generally consider the change in flow regime that occur when moving from the two-phase to a single-phase. The other three figures however are correct because. Over time the gas is spread over the pipe length resulting in the lower volume fraction, and given a relatively constant gas velocity the superficial velocity declines.
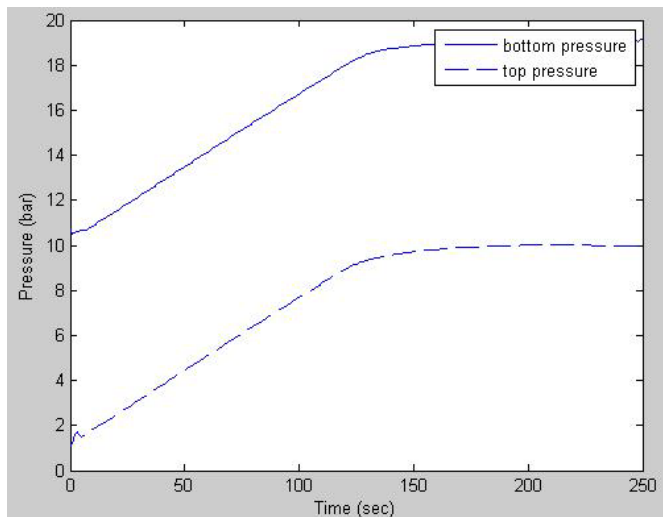
Figure 9

In figure 9 the pressure is displayed at the bottom and top of the pipe as a function of time. The pressure at both the bottom and the top is steadily increasing until the gas bubble reaches the top of the pipe. This phenomenon is because the liquid in the pipe is almost incompressible which can be seen from equation (27). Keeping in mind that the sound velocity in the liquid is 1000 m/s and reference density is 1000 kg/s, changes in pressure will not in this example alter the liquid density to a large extent. From the ideal gas law it is shown that:

$$pV = nRT \quad (103)$$

This means that if the temperature and volume are almost constant, the pressure in the gas bubble will be maintained during the rise in the liquid. When the gas bubble reaches the top of the pipe the pressure at the bottom will then be the pressure in the bubble plus the hydrostatic pressure from the liquid column. In the model this effect is taken in account in equation (28), which states that the density of the gas is dependent on the pressure.

Simulations

In this simulation the pressure, flow-rates and fluid velocities are going to be determined during a gas kick. Important trends in the results will then be compared against a similar case study in order to determine if the simulation produce the same trends for the different parameters.

The main goal in the simulation is to be able to produce graphs displaying; the pressure at the top and bottom, fluid inflow and outflow and fluid velocities, and then use these graphs to depict how these parameters behave during the gas kick and explain why these parameters changes the way they do.

Description of the case

The simulated case is motivated by a case study done by Avelar et al[3], and this case study will be used as a reference case against the simulated case. Initially drilling fluid will circulate down the drill string and back up the annulus, after a time a kick will be initiated at the bottom of the well. At this time the amount of liquid $Q_{L,Bottom}$ and gas $Q_{G,Bottom}$ flowing into the annulus at the bottom will be determined by the pressure at the bottom P_{Bottom}. The liquid flow out of the pipe at the top $Q_{L,Top}$ will be determined by the pressure at the top P_{Top}. When the pressure at the top reaches the shut in drill pipe pressure there will be no flow out of the well, however there will be an inflow of gas until the pressure at the bottom is in equilibrium with the reservoir gas pressure. The inflow and outflow parameters, pressure distribution and volume of gas in the pipe will be studied after the simulation is done.

To simplify the simulation in the program is will be based on two pipes rather than one pipe within another. The reason for this simplification is the fact that the frictional pressure drop is easier to calculate of these conditions. A sketch of the case will look like this:
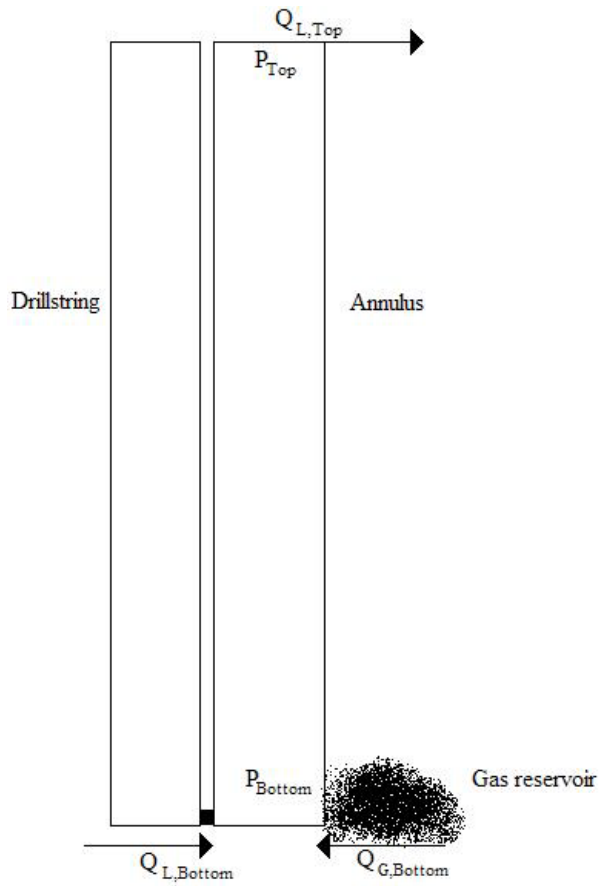
Figure 10

Data for the simulation is given in the following table:

Parameter	Unit	Value
Number of grids	N/A	50
Total time	s	1000
Max sound velocity	m/s	1000
Number of time steps	N/A	250
Viscosity of gas	Pa s	0,000005
Viscosity of liquid	Pa s	0,035
Length of pipe	m	500
Diameter of pipe	m	0,1
Sound velocity in gas	m/s	1000
Sound velocity in liquid	m/s	316,22
Reference pressure	Pa	100000
Reference pressure liquid	kg/m^3	1000
Shut it drill pipe pressure	bar	10

Table 3

Slip relation in this simulation is the same as the one used in the example and the calculation of the slip parameters K and S follow the equations (101) and (102).

The values in table 3 is not the same as the values use in the Avelar et al[3] case, however the main goal is to demonstrate similar trends as those observed by Avelar et al[3]. Getting the same results would require knowledge about what functions and parameters the case was based on. Rather the simulation will try to display some of the important phenomenon that occurs when there is a gas kick in a well. Since the model is based on mass flow per second rather than a volumetric flow rate, M as a notation of the mass flow rate per second.

The inflow and outflow of fluids are set at the start of grid one for inflow and the end of grind n for outflow.

For the flow of liquid out of the pipe the following relation is used:

$$M_{L,Top} = J_{L,Top} \cdot (P_{Top} - P_{Choke})^{nlt} \quad (104)$$

Where nlt and $J_{L,Top}$ are parameters depending on the flow and liquid parameters and the design of the outflow valve. The choke pressure at the top is set to 10 bar, which means that unless the top pressure is greater than 10 bar there will be no fluids flowing out of the pipe.

Relation for the inflow of the liquid is as following:

$$M_{L,Bottom} = J_{L,Bottom} \cdot (P_{L,Max} - P_{Bottom})^{nlb} \quad (105)$$

Again the parameters nlb and $J_{L,Bottom}$ are parameters dependent of the flow and liquid parameters and the design of the inflow valve. The maximum pressure available $P_{L,Max}$ in the equation is the pressure at the bottom of the drill string and can be expressed as:

$$P_{L,Max} = P_{Pump} + P_{Hydrostat} - P_{Fric} \quad (106)$$

Where P_{Pump} is the pump pressure on the rig, $P_{Hydrostat}$ is the pressure related to the hydrostatical pressure in the drill string and P_{Fric} is the frictional pressure drop in the drill string.

Finally the relation for the inflow of gas into the well is:

$$M_{G,Bottom} = J_{G,Bottom} \cdot (P_{Res} - P_{Bottom})^{ngb} \quad (107)$$

The parameters $J_{G,Bottom}$ and ngb are related to the flow and gas parameters and the reservoir parameters. After the bottom hole pressure reaches the reservoir pressure there will not be any more inflow of gas into the pipe.

Assumptions made of the simulation

Due to the fact that the program is already quite complex and might not respond well to other complex formulas for inflow and outflow in the pipe some assumptions where made in order to simplify the simulation. These however were made to promote stability in the simulations scheme and yet still show the desired phenomenons that occur. Values selected for the parameters in equations (104), (105) and (107) are displayed in the following table:

Parameter	Value	Unit
$J_{L,Top}$	$5*10^{-6}$	N/A
nlt	1	N/A
P_{Choke}	10	Bar
$J_{L,Bottom}$	10^{-6}	N/A
nlb	1	N/A
$P_{L,Max}$	60	Bar
$J_{G,Bottom}$	10^{-7}	N/A
ngb	1	N/A
P_{Res}	80	Bar

Table 4

The selection of nlt, nlb and ngb equal to one means that the mass flow is linearly dependent on the difference in pressure, one might argue that inflow and outflow of liquid is flowing through a valve and the flow should be dependent on the square root of the pressure. This argument can be derived for simplifying the Bernoulli equation and the resulting equation would be:

$$M = C \cdot A \cdot \sqrt{2\rho(P_1 - P_2)} \quad (108)$$

Where M is mass flow rate, C is the orifice flow coefficient, ρ is the density, P_1 is the downstream pressure and P_2 the upstream pressure.

However using the square root created tended to create oscillations in the model, where the outflow oscillated between flow and no flow for every other time step. The selection of a linear relation removed this effect without influencing the phenomenon to noticeable degree.

The selection of the $J_{L,Top}$, $J_{L,Bottom}$ and $J_{G,Bottom}$ should was based on a trial and error in the model. This mainly since the calculated values sometimes created oscillations in the mass flow rate particularly for the liquid phase. Ideally these values would be dynamic and dependent on their respective parameters. However keeping them constant did still display phenomenon in a correct way, and since the pressure variations are not very huge keeping them constant can be justified.

Keeping the pressure constant P_{Choke} can be justified since it is possible to control this pressure on the rig, P_{Res} will be constant unless there are dramatic changes in the reservoir and the $P_{L,Max}$ will depend slightly on the mass flow rate because of the frictional term that it contains. However for low mass flow rates setting the frictional pressure drop equal to zero in the drill pipe can be justified since it will not influence the model to a great extent, due to:

$$P_{Fric} \ll P_{Hydrostat} + P_{Pump} \quad (109)$$

This means that changes in the frictional pressure drop can be dismissed since it is a small percent of the total pressure.

Setup of the case in the program

MainStian

The changes in the main routine MainStian is to change the values of the geometry of the pipe to the desired values, the selection on which points to monitor N3 and N4 is also changed. The reason for changing the values of N3 and N4 is that if a higher numbers of grids are going to be used, the points will be automatically selected.

```
28> diam = 0.10;              %m
Area = pi*diam^2*0.25;

%
L = 500;

N1 = 1;
N2 = 5;
```

```
N3 = N-5;
N4 = N;
```

sol_stian

In order to make the mass flow out dependent of the pressure sol_stian must be changed to export the value of the pressure at the endpoint, p(n), to the massflow_out routine. This is simply done by adding the value in the following manner:

```
884> [massLout,massGout] = massflow_out(time,p(n));
```

init_funcC1_FL

Since the initial case in the model was a gas bubble rising from the bottom of the well to the top, the initial conditions must be changed to be only liquid. This is done by setting the initial gas fraction at the bottom equal to zero and the liquid fraction will then become one for the entire pipe.

```
131> % initial data
alpG_0(1,L)    = 0;
%alpG_0(1,~L1) = 0.25;
%alpG_0(1,L1)  = 0.999999;
%alpG_0(1,L1)  = 0.0;
alpL_0  = 1.0-alpG_0;
```

The length of the pipe is 500 meters, this means that the initial bottom hole pressure must be equal to the hydrostatic pressure of the liquid column plus the ambient pressure. The ambient pressure remained the same and the only change done is to set the length in this equation equal to 500.

```
175> pp = p_0 + C_rhoL0*g*(500 - x);
```

massflow_in

Mass inflow into the pipe is to be determined based upon the pressure at the bottom. For the liquid mass flow, the mass flow is based on formula (105), to avoid having a negative mass flow rate when the bottom hole pressure exceeded the $P_{L,Max}$ an if statement is used that set the liquid mass flow equal to zero once $P_{L,Max}$ is exceeded. The gas mass flow rate will start after a certain time, and after this time the mass flow rate will follow equation (107). Two if-statements are needed to make this function properly, one to make sure that there isn't any gas

flowing until after the time has passed and a second one within the first one to make sure there isn't any negative gas flow rates after P_{Res} has been reached.

```
13> if pin > 6000000;
    massL_in = 0.0;
else
    massL_in = (6000000-pin)/1000000;
end
end
if t>100
    if pin > 8000000
    massG_in = 0;
    else
        massG_in = (8000000-pin)/10000000;
    end
else
    massG_in = 0;
end
```

massflow_out

Top pressure isn't one of the input parameters in to massflow_out in the original simulation, and needs to be added. This is simply done by adding the "pout" as one of the parameters that massflow_out depends on.

```
1> function [massL_out,massG_out] = massflow(t,pout)
```

In order to make the outflow of liquid match the formula (104) and the assumptions made, an if-statement with an elseif-statement is needed. This is to make sure that there is no liquid flowing until the P_{Chocke} pressure has been reached and to make sure that once the shut-drill pipe pressure has been reach there will be no flow out at the top of the pipe, since this is the maximum pressure that is allowed until production stop. Gas flow out of the pipe is set to remain equal to one during the entire simulation.

```
11> if pout > 2000000
    massL_out = 0;
elseif pout > 1000000
    massL_out = 5*((pout-1000000)/1000000);
else massL_out = 0;
end
%massL_out = 0;
massG_out = 0.0;
```

PARAC1_FL

The PARAC1_FL is just some input parameters for the model, the only changes done in this routine is to change the amount of time that is simulated and the viscosity of the fluid in the simulation.

```
% #Noder  Tid      max sound velocity      NSTEP      muL    muG
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
  50      1000              1000              250       0.035   0.000005
```

Simulation

The main goals of the simulation are to study how the pressure at the bottom and at the top changes as the gas kick is under way, study the pressure gradient in the well, study the gas fraction in the well over time, volume of gas in the well of time, gas and liquid flow rate in and out of the well. The bottom hole pressure in the well must correlate to the bottom hole pressure in Avelar et al[3] simulated case and should look like this.
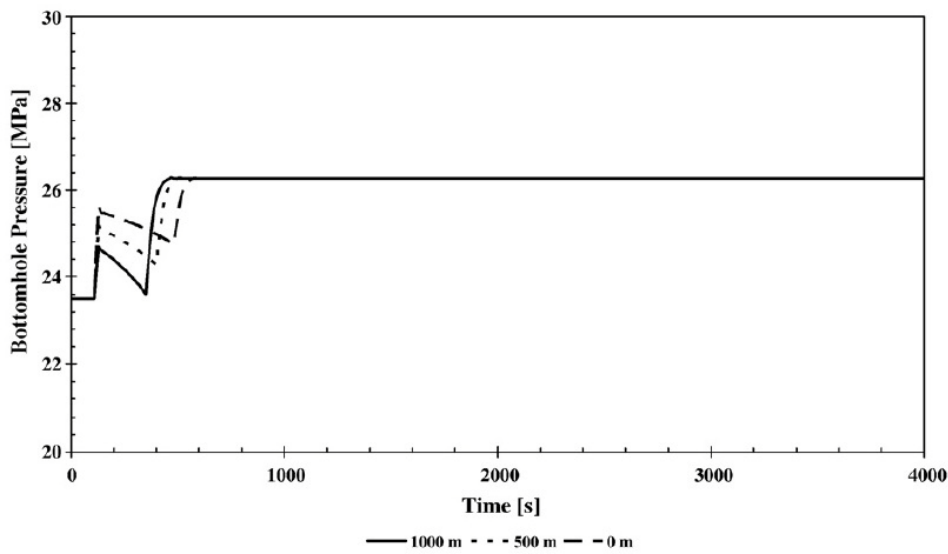
Figure 11

The graph for gas mass flow rate should have a shape like this:
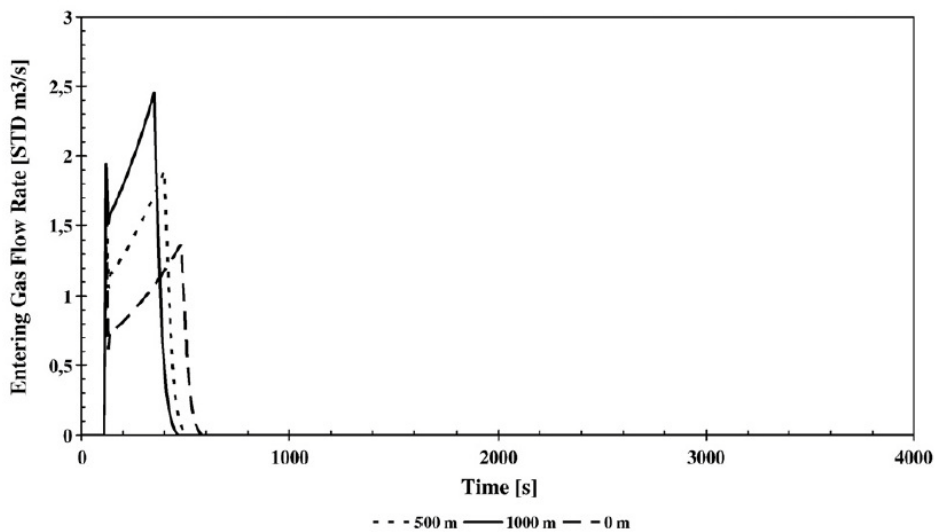
Figure 12

The graph corresponding to the liquid mass flow rate out of the well should have a shape like this:
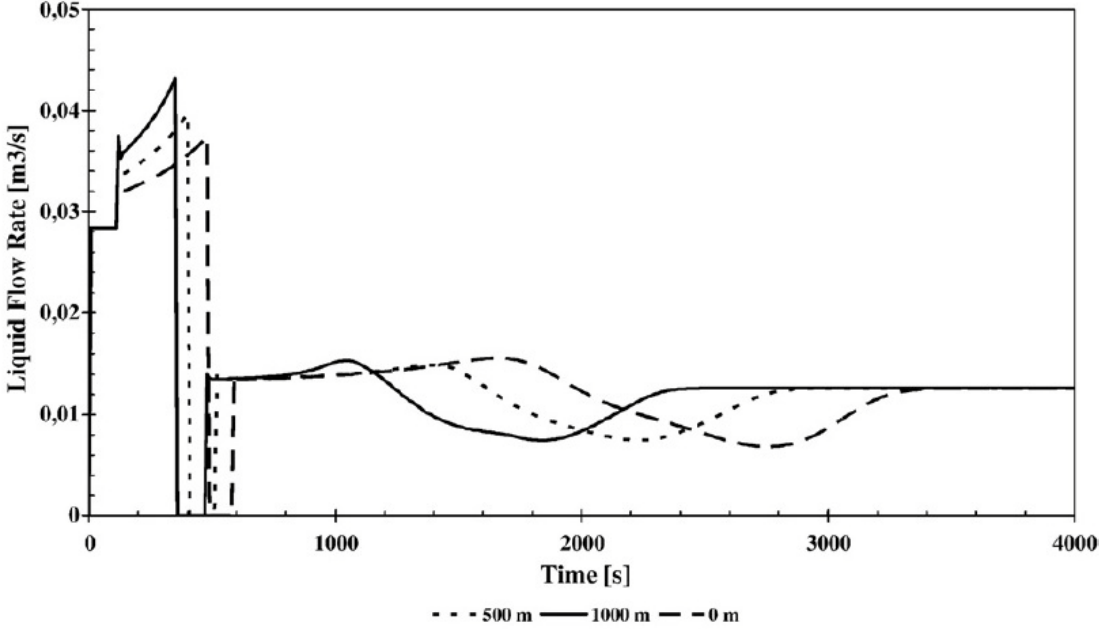
Figure 13

Results from simulations

Result of case simulation
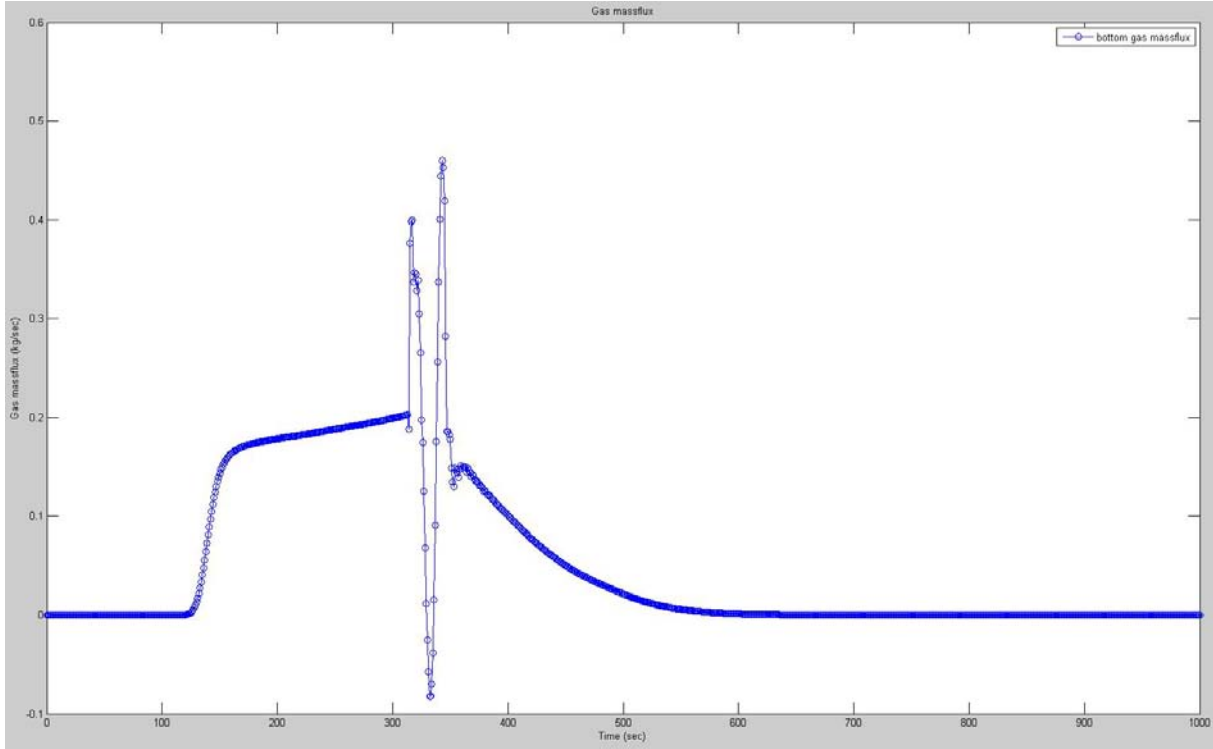
Figure 14

In figure 14 the mass flow of gas in to the well is plotted against time. The mass flow rate is given in kg/sec. For the figure the flow of gas starts after time t = 100 sec, since this is the point selected for the gas flow to start. Flow rate increases rapidly until an equilibrium point is reached and after this point the flow rate only increases slightly over time. Relatively large oscillations can be seen around time t = 300 sec. These large oscillations are due to the fact that mass flow of liquid is abruptly stopped at the top of the well. Despite the large oscillations the gas inflow rate seems to be simulated correctly since it diminishes quite rapidly after the top is reached, i.e. the liquid mass flow rate at the top is zero.
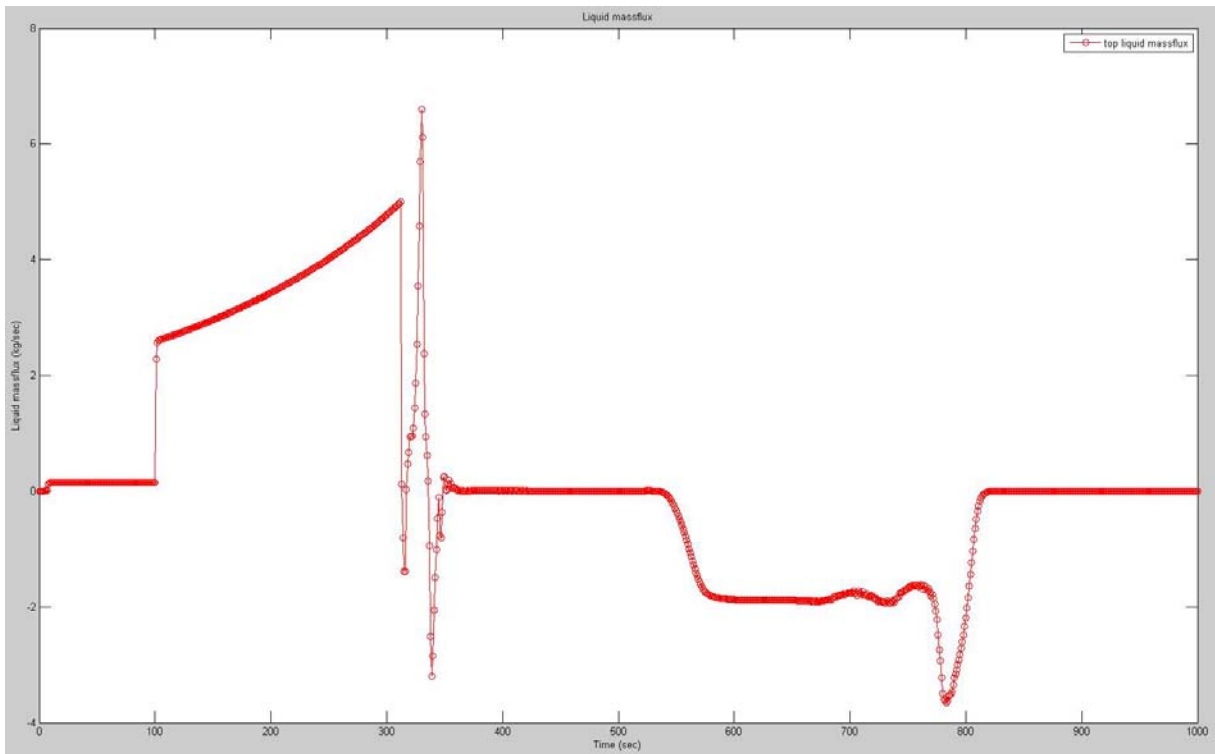
Figure 15

Figure 15 displays the mass flow rate of liquid out of the well at the top, the flow rate is measured in kg/sec. The flow rate is low and steady for the interval t = [0,100], but once the gas inflow start at the bottom of the well the liquid outflow rapidly increases due to the increase of pressure at the top of the well. A sort of equilibrium is rapidly reached, however after this equilibrium the flow rate of liquid still continues to increase at a high rate until the pressure at the top reaches the shut-in drill pipe pressure. When this pressure is reached the flow rate of liquid should go to zero, however due to some inelasticity in the simulation there is created huge oscillations of the pressure at the top and consequently large oscillations in the volumetric flow rate of liquid out the well. However once the oscillations have settled the flow rate remains zero. Since the flow rate displayed here is the change of volume in a cell very close to the top rather than the actual flow rate out of the well, there are some negative values of the flow rate once the gas reaches the top. This only means that the liquid have to move downwards in order for the gas to be able to reach the top of the well, since the gas is still moving due to buoyancy.
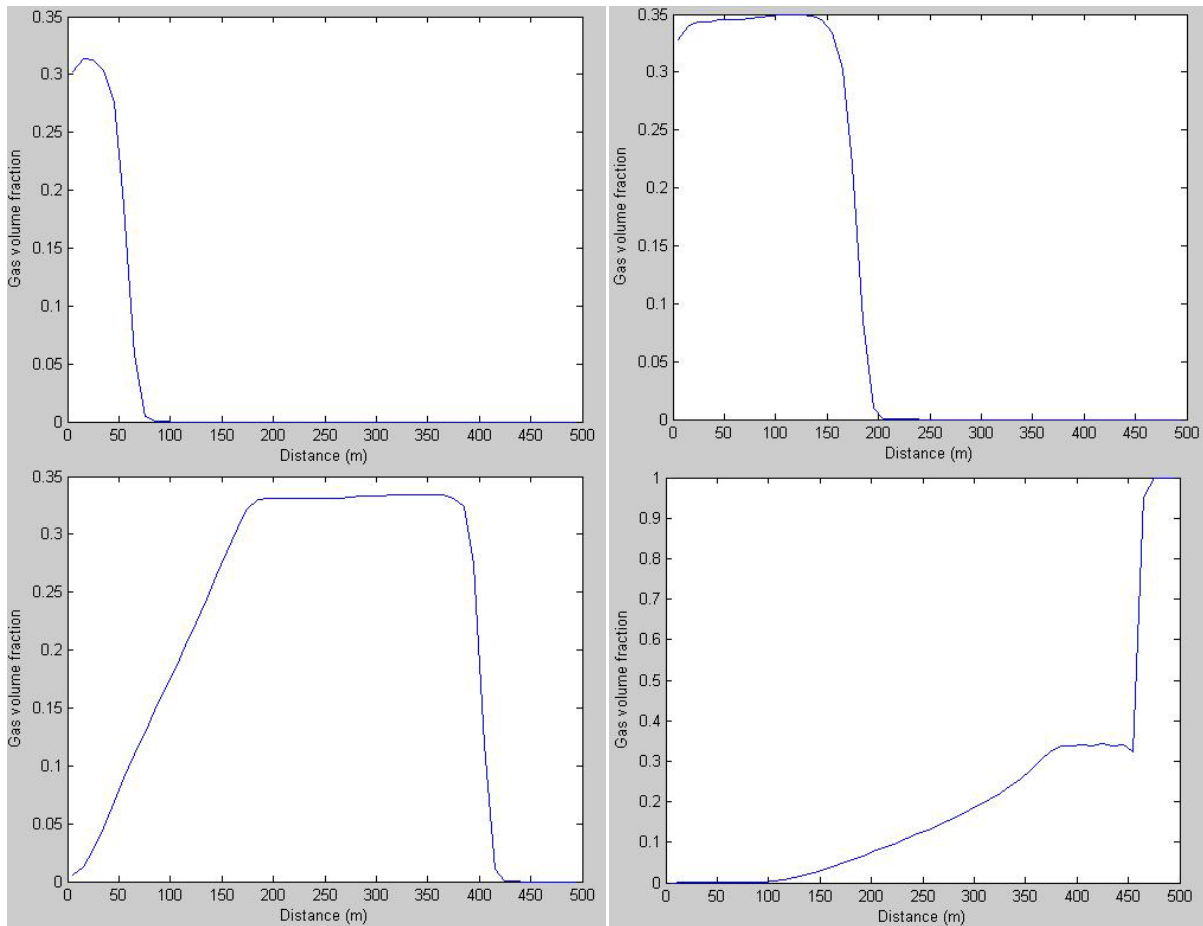
Figure 16

In figure 16 the gas fraction is depicted for the times t = 150 sec, 250 sec, 500 sec and 750 sec. For t = 150 sec the gas inflow has just started and the gas fraction is building up in the pipe, the front of the gas fraction is sharp due to the fact that the inflow of gas starts suddenly and increases fast until there is an equilibrium between the pressure difference and the mass flow rate. At t = 250 sec quite a bit of the gas has entered the well and the gas fraction has stabilized between 0.3 and 0.35, from the figure it can be seen that there is an increase of gas fraction higher up in the well, this is due to the fact that the gas is expanding as the pressure decrease higher up in the well. For t = 500 sec the gas inflow has stopped and the gas now only moves due to buoyancy, the gas fraction curve still has a sharp front but it has is smoother on the left side. The reason for this is that the mass flow of gas didn't stop very suddenly and the shape of the fraction is a mirror image of the mass flow rate of gas curve found in figure 14. After t = 750 sec some of the gas has reached the top of the well yet some is still flowing further down in the well. There is a unconformity in the around 450 meters this is due to a transition to single-phase gas regime which in turn makes calculation of velocity from the slip-law demanding.
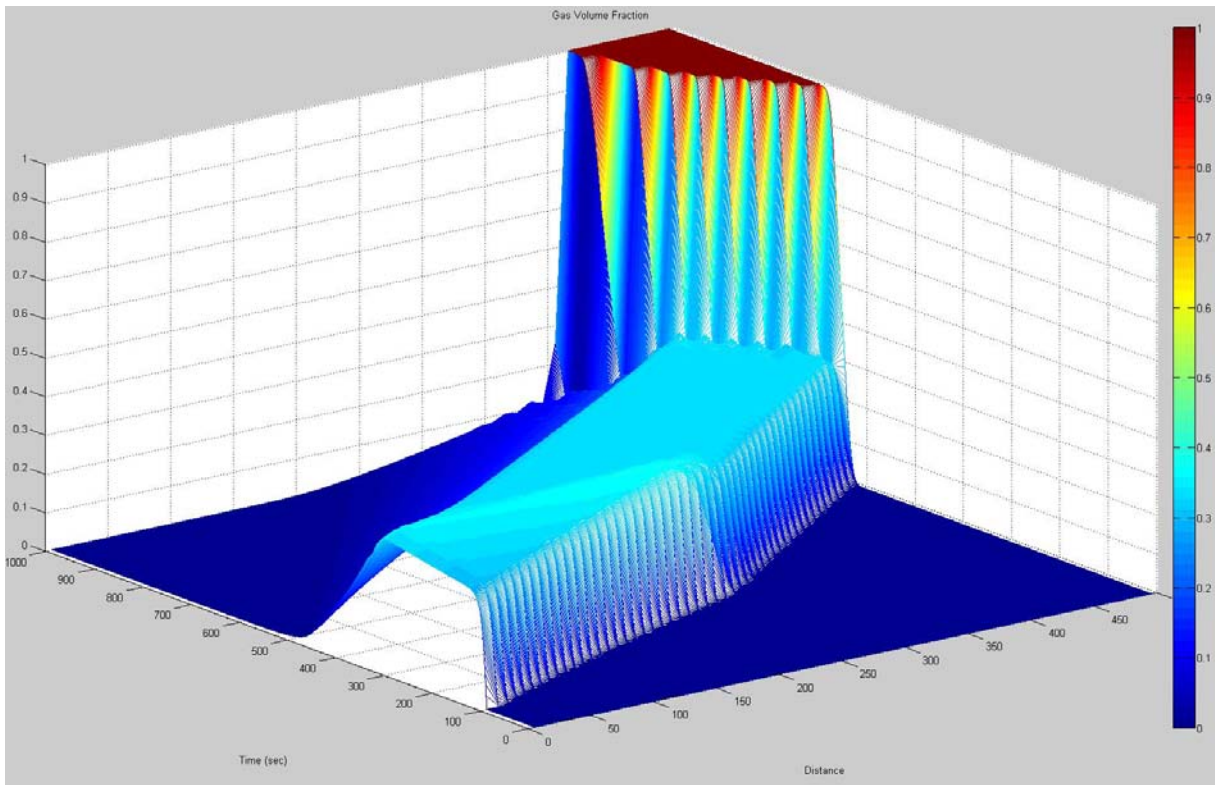
Figure 17

In the 3D figure, figure 17, it is easier to see how the gas propagates through the pipe over time. The gas moves with a sharp front, as can be seen from the right part of the figure and with a fairly constant value for the gas fraction.
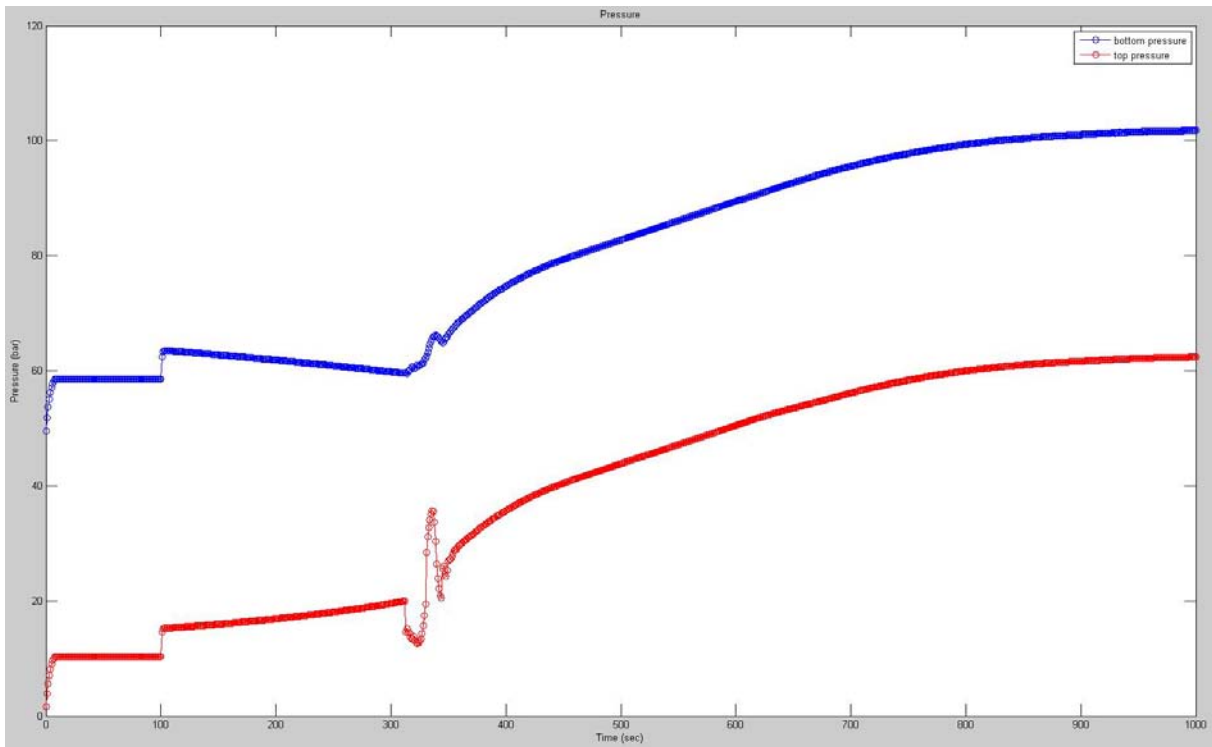
Figure 18

In figure 18 the pressure in bar is displayed for the top and bottom of the pipe, with blue indicating bottom and red indicating the top of the well. In the interval t = [0,100] the pressure at both the top and bottom is constant due to the restrains put into the model, i.e. maximum inflow pressure is 60 bar and minimum 10 bar to flow out. But when the gas kick starts pressure at both the top and the bottom suddenly increases since the reservoir the gas is coming from has a pressure of 80 bar. Beyond this point the pressure at the bottom decreases and at the top it increases. The reason it decreases at the bottom is due to the fact that gas in well lowers the hydrostatic pressure in the well and the frictional pressure drop in the well. At the top the pressure increases since the flow out of the well is dependent on the pressure at the top. The oscillation due to the closing of liquid outflow from the well at the top can also be seen in these graphs. However the oscillations at the bottom is much less than at the top, this due to the fluids in the well being viscose and absorbing much of the pressure oscillations before it reaches the bottom of the well. After the closing at the top the pressure at the bottom steadily rises until it reaches the reservoir pressure and the inflow of gas stops.
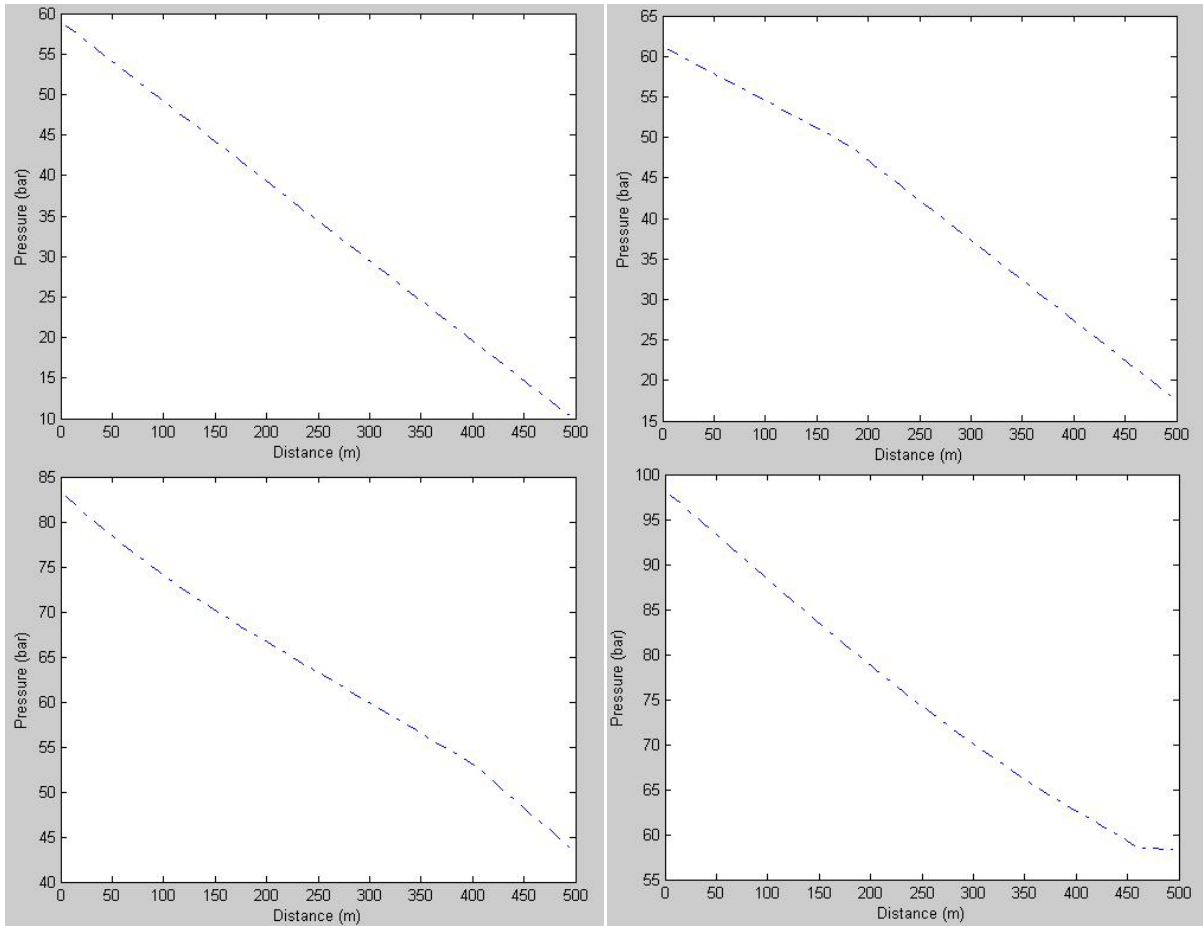
Figure 19

Shown in figure 19 is the pressure gradients for the well at times t = 50 sec, 250 sec, 500 sec and 750 sec. As expected when gas enters the well the gradient curve become flatter then when it is just filled with a liquid, since the mixed density of the fluid in the well is the fraction weighted average of the liquid and gas density. When the gas bubble reaches the top the gradient is almost a flat line, meaning that the pressure from lower down in the well is maintained to the top of the well. Additionally it is seen in that for t = 750 sec the pressure difference between top and bottom is below 40 bar, compare to 50 bar of the liquid column at t = 50 sec.
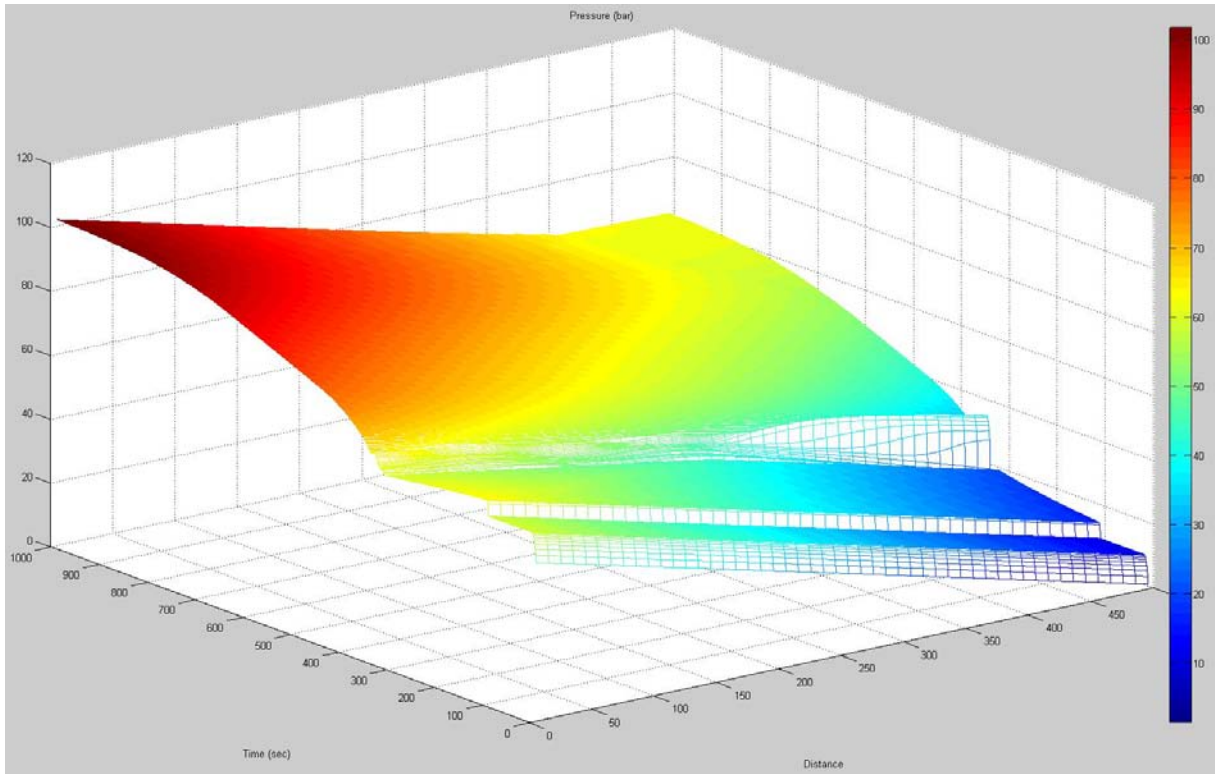
Figure 20

Figure 20 merely gives a 3D representation of the phenomenon shown in figure 19. The two "jumps" on the figure represent the start of the gas flowing in to the well and the closing of the well at the top.
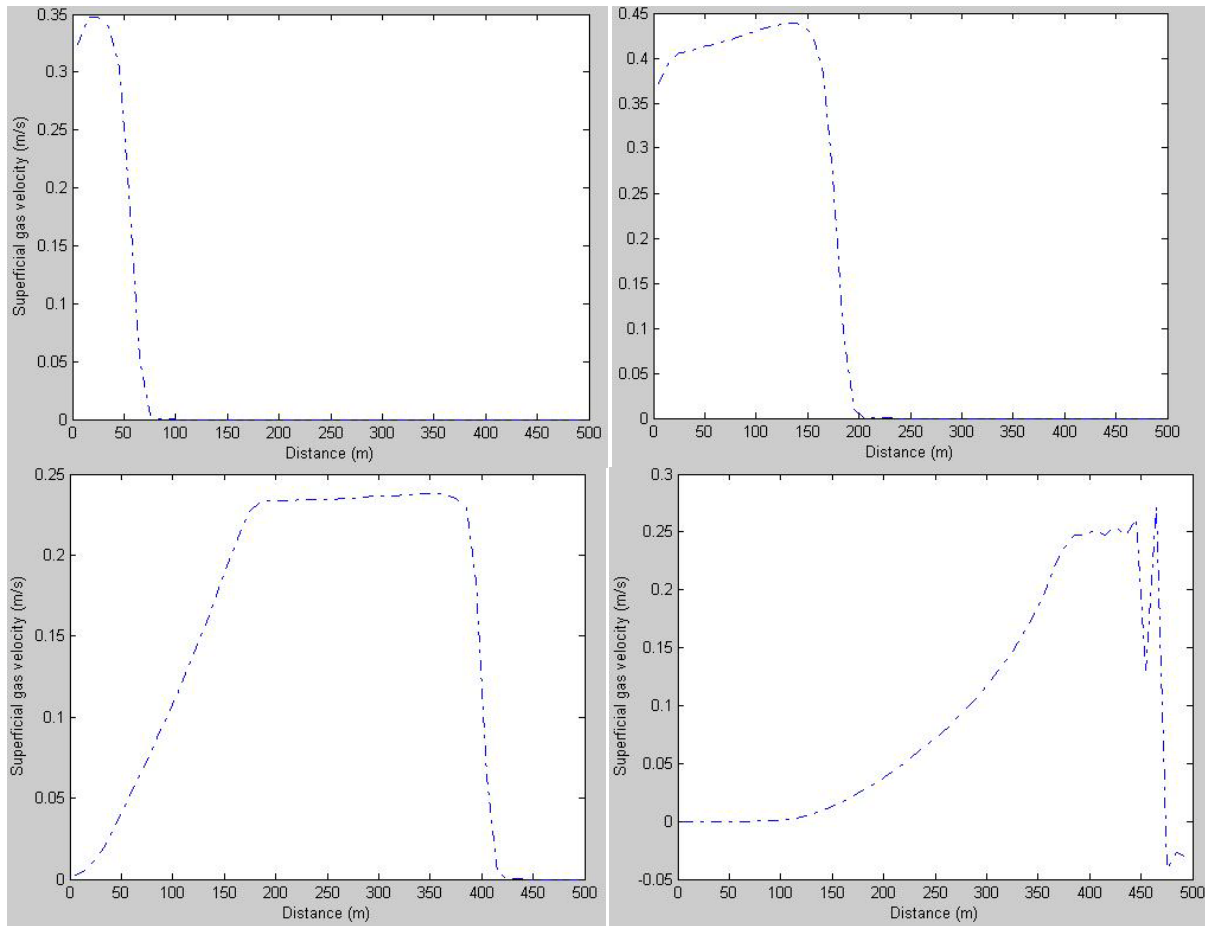
Figure 21

The figure 21 displays the superficial velocity of the gas in the well. The graphs are very similar to the graphs for the gas volume fraction, figure 16. The reason is, as seen in formula (7) and the slip assumptions in formulas (15), (17) and (19), that the superficial velocity is dependent on the gas fraction, mix velocity and terminal rise velocity. Hence when the liquid stop flowing the superficial velocity of the gas decreases, this is seen from the values on the y-axis on the graphs from t = 250 sec against t = 500 sec. Again it is seen that when there is a transition to single-phase gas flow there is a problem selecting slip parameters such that a physical solution is obtained.

Following figure 23 and 24 give a 3D representation of the velocity that the gas and liquid has during the simulation. The liquid velocity increases abruptly when the gas inflow is introduced, and when gas is flowing in the same grid cell as the liquid the liquid has a velocity close to zero, since the gas flowing up displaces the liquid forcing it to flow downward.
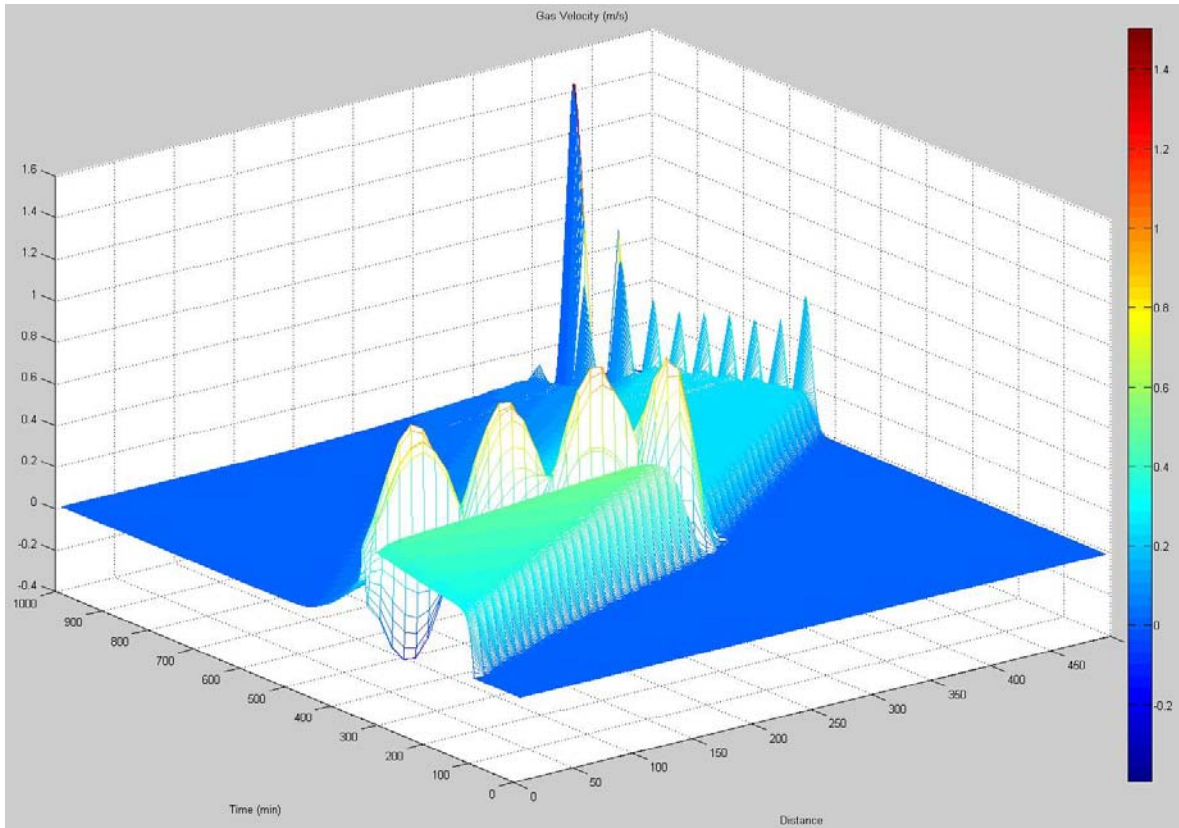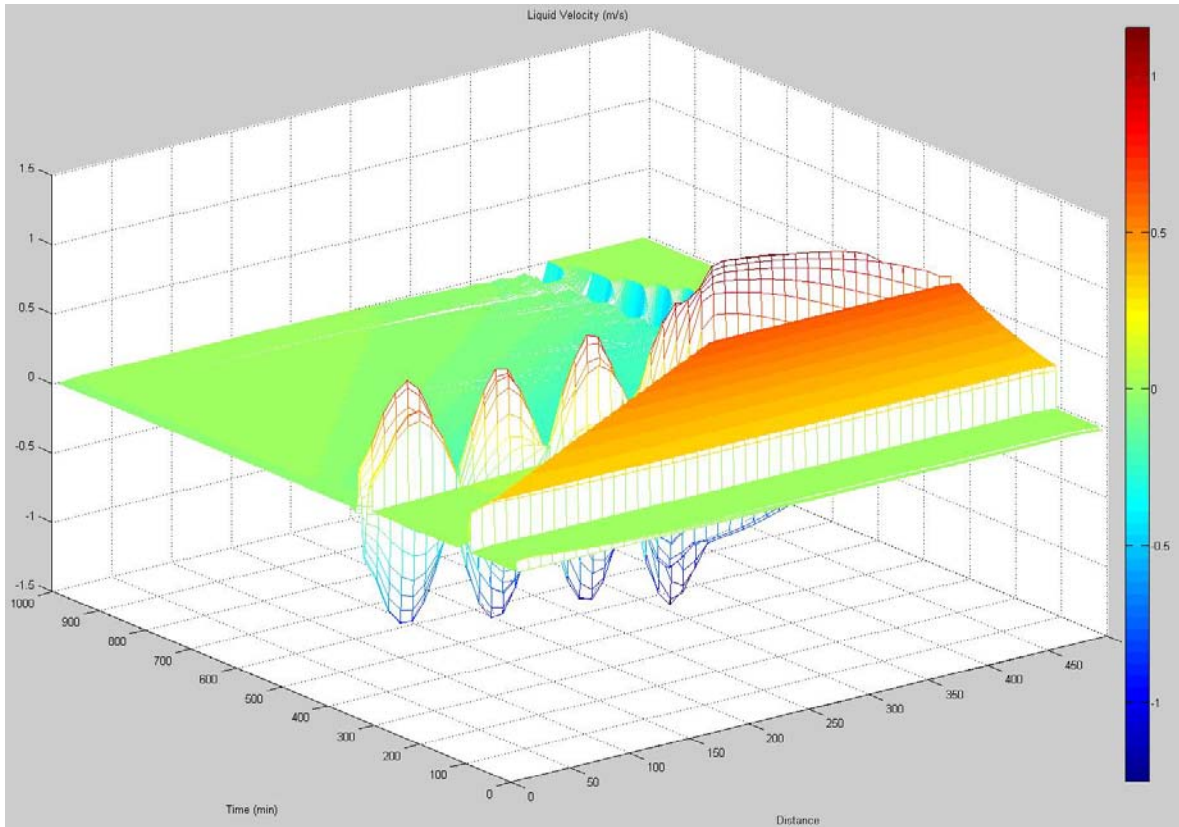
Figure 22

Figure 23

Discussion

Pressure

Comparing the bottom-hole pressure in Figure 11 and Figure 18, it is seen that the curves have the same shape and illustrate the same phenomenon. Immediately following the gas inflow the bottom-hole pressure increases. The increase in pressure can be attributed to the fact that there is an overpressure in the reservoir compared to the pressure at the bottom of the well. These two pressures will then try to form equilibrium of pressure and gas inflow, as seen in formula (107).

In both cases the pressure declines once the gas has entered the well. When the gas has entered there are several effects of the fluid in the well. The mixture density, viscosity and sound velocity is all lowered. Changes in the sound velocity will only affect how fast the pressure changes propagate through the well. Considering formula (29) it is seen that when the gas fraction increases in a grid cell, the mixture density is lowered and as a result the hydrostatic pressure is lowered across the grid cell. In formula (30) the relation for the frictional pressure drop is shown and from the formula it is seen that since the gas lowers the mixture viscosity the frictional pressure drop will be lowered. These two mechanics combined give the lowering of the bottom-hole pressure in the well after the initial equilibrium that was formed.

The pressure is decreasing in until it reaches a bottom value. This bottom value signals that the outflow of liquid from the well has been stopped, and when there no longer is anything flowing out of the well the result is a steep increase in pressure at the bottom. This steep increase is better seen the reference case (from Avelar et al[3]) then from the simulated case. The reason that this is seen as a much sharper increase in the reference case then in the simulated case is probably the fact that they selected other conditions for closing the well than was done in this simulation. Selecting other closing condition will remove the oscillations of pressure in the well when the liquid outflow is halted. The simulated case could also become more equal to the reference case by increasing the time of the simulation which would yield the long flat line seen in figure 11.

The reason that the pressure increases once the liquid outflow is stopped, is due to the well being closed at the top and the liquid in the well is nearly incompressible. The gas traveling up the well will "carry" the pressure from lower down in the well. Together with the fact that the pressure from the reservoir has to create a different equilibrium with the bottom-hole

pressure, results in the steep increase of the bottom-hole pressure after the outflow has been stopped.

Towards the end the inflow of gas is greatly reduced and the velocity of the gas traveling up the well is only corresponding to the terminal rise velocity. The bottom-hole pressure then slowly moves towards an equilibrium which is kept. (30)

Inflow

The graphs for the inflow of gas into the well can be seen in figure 12 for the reference case and figure 14 for the simulation done. The reference case uses volumetric flow rate while in the simulation mass flow rate was used. Since the pressure at the bottom isn't constant it can be seen form formula (28) that the density of the gas will vary. This means that the shape of the curves will not be equal especially at the end of the gas inflow, since the pressure increase at the end is very large, from around 60 bar to around 100 bar. This and the fact that the inflow conditions selected are different means that the figure vary from each other, they do however both depict the interesting phenomenon that occur during the gas kick.

After the gas kick starts the gas inflow rate quick increase to the value that corresponds to the pressure equilibrium in formula (107). Following this equilibrium the pressure at the bottom decrease as explained previously, and to maintain the equilibrium between inflow and bottom-hole pressure the gas inflow rate continue to increase. At this point there is an outflow of liquid at the top, this means that the gas velocity is determined by both the terminal rise velocity and the mixture velocity of the fluids.

Once the liquid outflow ceases there is a steep increase in pressure at the bottom and the velocity of the gas is only equal to the rise velocity of the gas. Still following the equilibrium in equation (107), the gas inflow rate rapidly decreases until the bottom-hole pressure is equal or greater than the reservoir pressure.

The oscillations cause by the rapid shut in of the well at the top can also be seen in the graph for the simulated case. Again these oscillations are seen in the inflow since the inflow is related to the bottom-hole pressure. However these values will not be as severe as the oscillations is being dampened by the viscous fluids in the well.

Outflow

Comparing the liquid outflow rates in figure 13 for the reference case and figure 15 for the simulated case, the only relevant part is the first part until the well is shut-in. This is because the well in the reference case opened the well again after some time in order to circulate out the gas in the well. As for the gas inflow the reference case uses volumetric flow rate while the simulated case uses mass flow rate, however this will not yield significant differences in the two figures since the liquid is almost incompressible.

Initially there is a steady flow of liquid out of the well, this corresponds to normal drilling where mud is circulated through the well in order to remove the drill cuttings at the bottom, cool the drilling bit and lubricate the system. The liquid flowing out of the pipe follows the relation given in equation (104) and the constraints given in table 3.

When the gas kick is initiated the pressure at the top also increase, this means that in order to maintain the equilibrium the outflow of liquid increase according to the formula. This is seen in both the cases.

While the gas kick is under way the pressure gradient in the well is lowered, as seen in figure 19. This means that although the pressure at the bottom of the well is decreasing the pressure at the top is increasing. Following the relation between the top pressure and the liquid outflow, the liquid outflow rate steadily increases until the shut-in drill pipe pressure is met. Once this pressure is met the liquid outflow at the top is halted to avoid filling the whole well with gas and getting the full reservoir pressure at the top.

The outflow rate oscillations are more severe for the liquid than for the gas since there is no column of fluid to dampen the pressure, this means that liquid outflow oscillates until the model reaches its equilibrium. This can be prevented in the model by selecting a less strict closing condition.

Conclusion

The results from the simulation done match the result from the reference case quite well, considerer that the simulation was done with no knowledge of the inflow and outflow functions used and how slip was defined in the reference case. However the most important aspect is that the results from the simulation displayed all desired phenomenons in the well during the gas kick.

The model has some difficulty, as expected, to simulate the transition from two-phase to one-phase. Simulating this transition is an inherit problem when using the drift-flux simplification in a model. This is because the slip parameters seen in function (26) will not behave in a manner like this when the gas fractions get high. Especially when the gas transitions from a two-phase to a single-phase at a steady rate, like at the end of the simulation, the gas fraction will be correctly determined in the low fraction area and in the single-phase area, however in-between the calculations are inconsistent with these two "end points". This is displayed in figure 16 and figure 21.

The other artifact in the simulation is the oscillations when the liquid outflow from the pipe is halted. To negate this from happening there should have been implemented a less strict cut of point or the liquid outflow should have been defined at zero after the well pressure reached the shut in pressure once. This would have removed if not all but some of the oscillations, but since the oscillations them self showed how the pressure waves would be dampened as they move down the well it was kept in the model.

Calculation time for the simulation is also relatively low despite having a high number of grid cells and time steps. However due to the implicit nature of the model, it is not able to gain any benefit from having several calculation threads running at the same time since the model is only able to use one of them. If several threads available for calculations, it is possible to run several simulations at once without slowing down any of the calculations. This was especially well seen when running the model on a quad-core processor, where the MatLab CPU usage peaked at 25%.

Despite have some limitations in terms of predicting the transition from two-phase to single-phase for the gas and the limitations in the model in terms of thread usage on a CPU, the model is very versatile in that one is able to select and define almost every parameters for a well and still get results that are consistent with what one would expect. Therefore this model can easily be implemented to simulate a wide specter of cases and still come up with correct answers.

Reference list

[1] R. W. Time. "Two-Phase Flow in Pipelines" University of Stavanger 2009

[2] S. Evje and K. K. Fjelde. "Hybrid Flux-Splitting Schemes for a Two-Phase Flow Model" Journal of Computational Physics Volume 175 Issue 2, January 20 2002

[3] C.S. Avelar, P. R. Ribeiro and K. Sephrnoori "Deepwater gas kick simulation" Journal of Petroleum Science and Engineering 67 (2009) 13-22

[4] Y. Wada and M. Liou "An accurate and robust flux splitting scheme for shock and contact discontinuity" SIAM Journal on Scientific Computing 18(3) 633-657, 1997